OPEN*control*

End User Manual

DOCUMENT NUMBER:	4602H
RELEASE REF.:	
EDITION:	00
AUTHOR:	**Prima Electro S.p.A.**
DATE:	**June 2013**

PRIMA ELECTRO reserves the right to modify and improve the product described in this manual at any time and without prior notice. The application of this manual is under customer responsibility. No further guarantees will be given by PRIMA ELECTRO, in particular for any possible faults, incompleteness and/or difficulties in the operation. In no event will PRIMA ELECTRO be responsible or liable for indirect or consequential damages that may result from the use of such documentation.

COPYRIGHT 2013- Prima Electro S.p.A.
All rights reserved. Reproduction, use or disclosure to third parties without express authority is *strictly forbidden*.

Edition	Date	Notes	Author
00	14/06/2013	First DRAFT	Prima Electro S.p.A.

Contents

1. Features and specifications 7

1.1 Reading keys and security instructions 7

1.2 General 7

1.3 Hardware structure 8

1.3.1 Operator Panel 9

1.3.2 OPENconsole and OPENconsole COMPACT 9

1.4 WinNBI Interface Software 10

1.4.1 Connection between the WinNBI and the CNC 11

1.4.2 Common features of the WinNBI applications 12

2. System Start-up 14

2.1 System Power Up 14

2.2 System Shut Down 14

2.3 Diagnostic Screens 16

2.4 Switching the System in Special Operating Modes 17

2.5 Selection of the Operating Mode 18

3. User Interface 21

3.1 The Screen 21

3.2 The Keyboard 21

3.3 The Control Panel 21

3.4 The ProcessController 22

3.4.1 Title bar 23

3.4.2 Menu bar 23

3.4.3 Tool bar 23

3.4.4 Operator Panel bar 24

3.4.5 OEM Operator Panel 25

3.4.6 Screen layouts 26

3.4.7 SoftKey bar 26

3.4.8 Main video screen osai_p6a.qv 27

3.5 CNC Status Area 28

3.6 Flags Area 30

3.6.1 Axes positions and origin area 31

3.6.2 Feed and overrides Area 32

3.6.3 Selected Tools Area 33

3.6.4 G and M functions Area 34

3.6.5 Part program data area 34
3.7 Keyboard Commands 35
3.7.1 Additional information windows 35
3.8 Panels for Command and data input 36
3.8.1 General rules on the input windows 36
3.8.2 The ProcessController soft-keys 37
3.8.3 Multiple connection, CNC selection 40
3.8.4 MDI Panel functions 54
3.9 Operator Consoles 55
3.9.1 Keys and functions of the OPENconsole COMPACT Operator Panel 55
3.9.2 Keys and functions of the OPENconsole COMPACT Operator Panel 57

4. Establishing origins and homing the axes 58

4.1 General 58
4.2 Axis homing procedure 58
4.3 Definition of the origins 59
4.3.1 Manual setting of the absolute origin 60
4.3.2 Automatic set of the absolute origin 61
4.4 Axes and part origins 64

5. Axes jogging and Stop functions 66

5.1 JOG 66
5.1.1 Continuous jog 67
5.1.2 Incremental JOG 68
5.2 Manual Feedrate change 68
5.3 Hold entry and JOG return on the profile 68
5.4 Spindle Speed Override 70
5.5 Stop functions 70
5.5.1 Reset 70
5.5.2 Hold 71
5.6 Emergency Stop 71
5.7 Active reset 71
5.8 Removing the tool after an Emergency 74

6. Part Program Management 77

6.1 Main Video Screen 77
6.2 Selection of an item 78
6.3 Multiple selection 78
6.4 Creation of a new Part Program 79
6.5 Creation of a new Directory 79
6.6 File copy and directory 80

6.7 Delete a directory or a file ... 80
6.8 Change the name of a file or a directory ... 81
6.9 Writing and editing a Part program ... 81
6.10 Enabling a Part-Program for the execution ... 82
6.11 Disabling the Part-Program selected ... 82
6.12 Program Management Configuration ... 83
6.13 Logical drive configuration ... 83
6.13.1 Adding a logical drive ... 84
6.13.2 Removing a logical drive ... 84
6.13.3 Showing logical drive path ... 84
6.14 Four Windows Display (Quad View) ... 85

7. Part Program execution ... 86

7.1 Part program execution ... 87
7.1.1 Selecting and activating a part program ... 87
7.1.2 Automatic execution ... 87
7.1.3 Block-by-block execution ... 87
7.1.4 Multiblock retrace ... 88
7.2 Executing blocks from the keyboard ... 89
7.2.1 MDI execution without an active part program ... 89
7.2.2 MDI execution with an active part program ... 89
7.2.3 Executing part of a part program ... 90
7.3 Modifying blocks in BLK/BLK mode ... 90
7.4 Strings search ... 91
7.5 Feedrate Override ... 91
7.6 Spindle feedrate override ... 91
7.7 Rapid Feedrate Override ... 92
7.8 Execution in *Dry Run* mode ... 92
7.9 Memory Search ... 92
7.9.1 Automatic Search ... 93
7.9.2 Search for a pre-set block ... 93
7.10 Restart a working cycle ... 94
7.11 Searching Mode ... 94

8. Using Tables ... 96

8.1 Enabling Table Editor ... 96
8.2 Origin Table ... 102
8.3 Tool Table ... 104
8.4 Tool Offset Table ... 106
8.5 User Table ... 110

9. The Table Builder 111

9.1 General 111

9.2 Origin table customization 114

9.3 Tool Table customization 119

9.4 Table Offset Customization 124

9.5 User Table customization 129

9.6 Translation of the variable names 136

9.7 Configuration file 137

10. Tool Management 138

10.1 General 138

10.2 Tool Pre-setting 138

10.2.1 Manual Tool pre-setting 138

10.2.2 Manual pre-setting of the tool length mounted on the spindle 141

10.2.3 Automatic pre-setting of the tool mounted in the spindle 142

11. Machine Plot 143

11.1 General 143

11.2 Machine Plot Interface 143

11.3 Machine Plot Configuration 147

11.4 Profile Drawing 151

11.5 Drawing a 3D Profile 153

11.6 Checking the Profile 154

11.7 Drawing the disconnected axes profile (DRY RUN) 157

12. System History 158

12.1 General 158

12.2 System History Interface 158

13. Appendix A 164

13.1 Error Messages 164

1. Features and specifications

1.1 Reading keys and security instructions

ATTENTION! To use the system in a correct manner, follow the instructions provided in this manual and pay special attention to the symbols below:

WARNING! Dangerous Voltage: This symbol is associated with dangerous high voltages that could damage the system, the equipment and the operators.

WARNING! Danger: This symbol is associated with facts and circumstances that could damage the system, the equipment and the operators.

NOTES: This symbol is used for operations that have to be executed with great care in order to ensure their successful completion.

1.2 General

OPEN*control* is a family of state-of-the-art numerical controls, designed to meet a vast range of standard and non-standard application requirements. The most powerful model can manage up to 64 digital axes. The OPENcontrol systems can be installed on many types of machine: milling machines, grinding machines, wood-, glass- and marble-working machines, oxygen cutters, etc. Thanks to the "open" Hardware and Software architecture, the systems are particularly suitable for both standard and for those applications requiring special functions. OPEN*control* allows the manufacturer to customize the system to integrate with their specific application requirements.

The OPEN*control* systems are based on the most advanced technologies; from the hardware components to the powerful Operating System (multitasking, real time, event driven) complete with either a local or networked graphic interface, that makes the OPEN*control* system one of the most important CNCs. The powerful and sophisticated functions typical of numerical control, can be combined with a user friendly interface, configurable according to the MTB's requirements. This interface is named "WinNBI" (Windows Network Based Interface) and is displayed on the operator panel connected to the CNC.

There are several HMI types to match with the OPEN*control*, from the simple industrial monitor up to the more advanced Operator Panel. This manual describes the user interface provided by the OPEN*console* Operator panel and the WinNBI application graphical software. In the following pages, there is an overview of the configuration to which this manual refers.

For any additional information and a complete evaluation of all the OPENcontrol hardware configuration, refer to the manuals below:

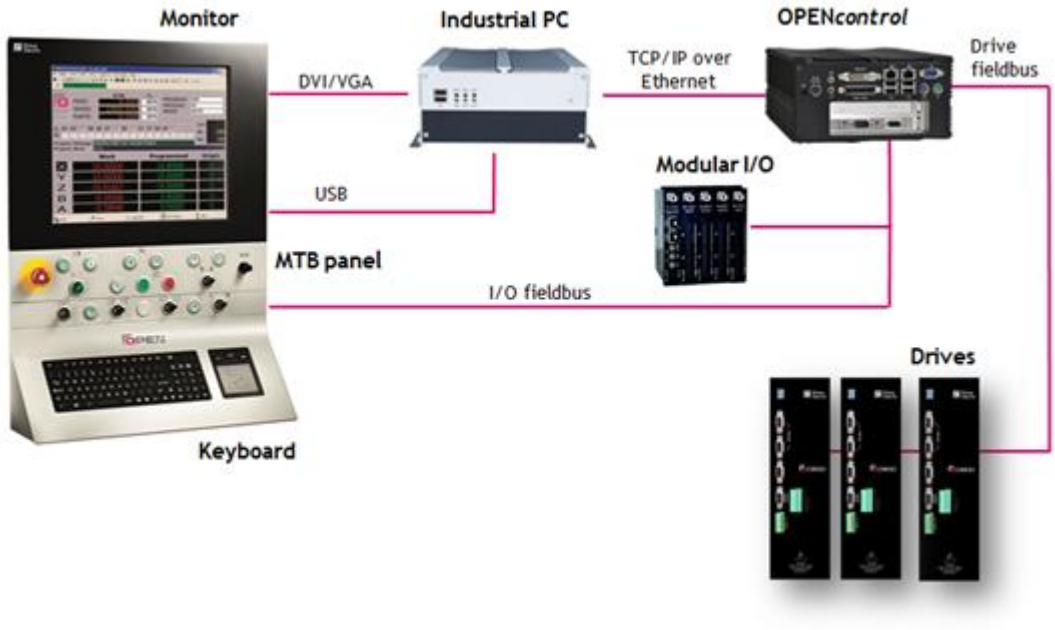
- "OPEN*control* – Software Installation Manual" code 45006692G.
- "OPEN*control* – Installation Guide" code 45006652F.

1.3 Hardware structure

The OPEN*control* systems are made up of a number of modules that can be combined in different ways according to the requirements. Usually, the system has an operator panel used to display the data and to handle commands to be sent to the CNC.

Only skilled operators together with the MTB can open the operator panel and the basic unit for “service” reasons.

This manual explains how to use the OPEN*control* CNCs combined with the OPEN*console* operator panel as shown below.



In this configuration the OPEN*control* Control Unit is connected on the network to an industrial PC dedicated to the graphical user interface. The control unit is therefore dedicated to the real-time software, while the data is displayed on the industrial PC on the OPEN*console* Operator Panel. The other devices, such drives and I/Os, are connected to the Control Unit.

In other configurations, the user interface software runs on the control unit without using an industrial PC between the control unit and the operator panel.

This manual describes the use of standard video screen layouts provided by Prima Electro with the product software release. The structure of the video screen layouts can be changed by the MTB, and may therefore appear to the end user in a different way. In this case, the MTB will describe the functionality of the customized screen layouts.

1.3.1 Operator Panel

The Operator Panel (or front panel) is the interface device between the system and the operator.
In all versions, the Operator Panel is equipped with a LCD monitor, a standard USA-ASCII keyboard with function buttons that work as softkeys.
It offers the most up to date Man-Machine interface techniques together with ease of use, its main functions are:

- introduction of operating commands.
- introduction of start, stop and reset commands.
- the data input using the keyboard
- software and data entry (part program, software options, etc.), using USB connected devices.
- display of the data and all the system operating conditions.
- control of the CNC and PC ambient.

This manual deals with the OPEN*console* operator panel use and the WinNBI graphical user interface by Prima Electro.

1.3.2 OPENconsole and OPENconsole COMPACT

The OPEN*console* operator panel offers great modularity and an innovative design. The panel consists of independent modules that can be assembled together to obtain the ideal configuration match the specific application requirement. The Control Panel exchanges data using an EtherCAT fieldbus. The panel is equipped with a silicon rubber keyboard, Windows standard, that is supplemented by a Touch Pad or Track-Ball unit. The 17" or 19" Monitor, has a resistive integrated Touch Screen. Both the monitor and the keyboard can be connected directly to the CNC or to the interface PC (if this is provided in the system configuration).

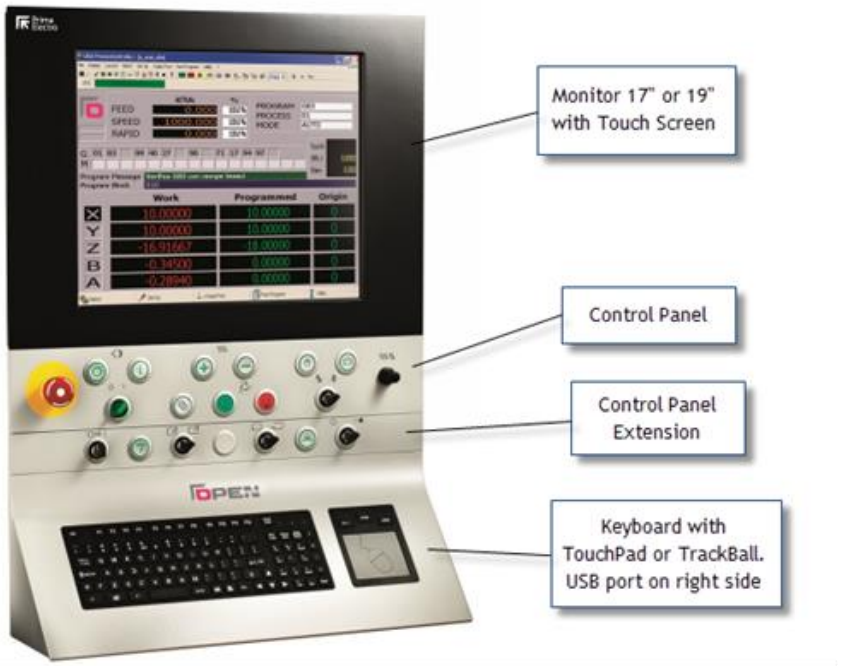


Figure 1: OPENconsole configuration with Monitor, Control Panel, Control Panel Extension and Keyboard with TouchPad

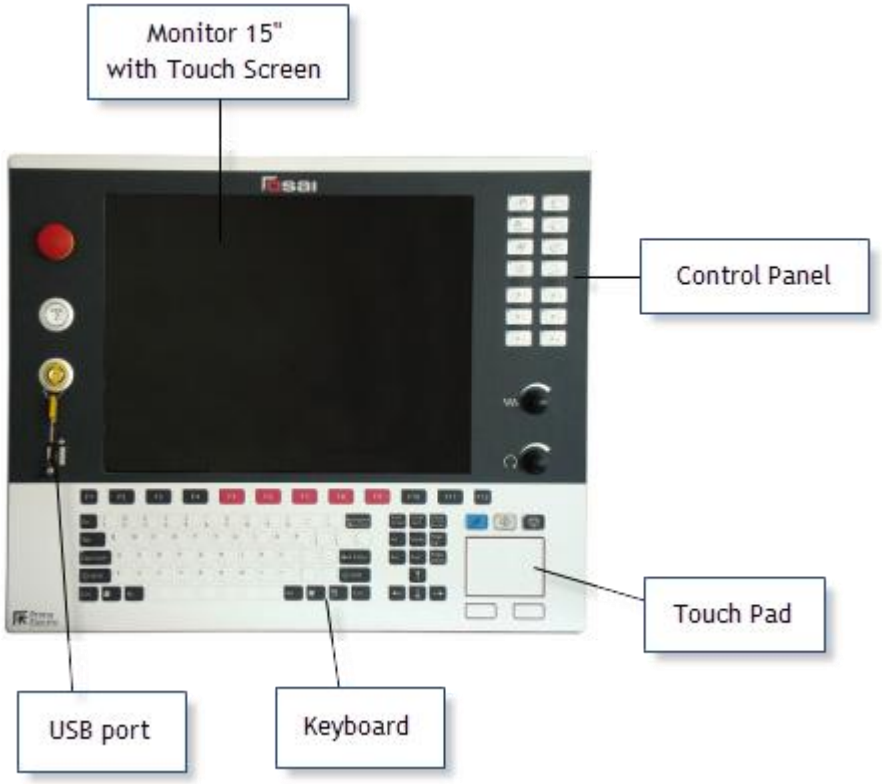


Figure 2: OPENconsole COMPACT - all HMI elements in one module

1.4 WinNBI Interface Software

The WinNBI graphics suite (Windows Network Based Interface) consists of several different applications or independent HMI utilities, each dedicated to a specific function. These applications allow the complete management of the machine tool. The applications (standard or optional) are for users having different requirements (end user, MTB, service engineer). This manual describes the WinNBI utilities used by the end user:

ProcessController Data display and commands for the CNC.
This is the main WinNBI application which displays all the CNC data: axes position, speed, active functions, program lines etc. and also allows the sending of commands and parameters to the CNC. The Process Controller is supplied with some standard video screens to manage the machine in manual and automatic mode. In addition the optional Layout Builder ambient allows the screen content to be adapted to suit the exact requirements of the application

Therefore, the ProcessController can present a graphic interface completely different from the standard one supplied and documented by Prima Electro.

BootController Numerical Control boot controller.
It displays the messages from the CNC during the boot stage. It also allows the system software restart and setting of the operating state that the CNC must be in for machining, tuning and service.

TableEditor Data table editor

It allows display, entry and modification of some parameters, in particular:

- origins
- tools
- tool offsets
- user tables
- tool magazines
- tool data base

The TableBuilder is part of the TableEditor allowing customisation of the data and the data display within the same tables. For details see chapters 8 and 9.

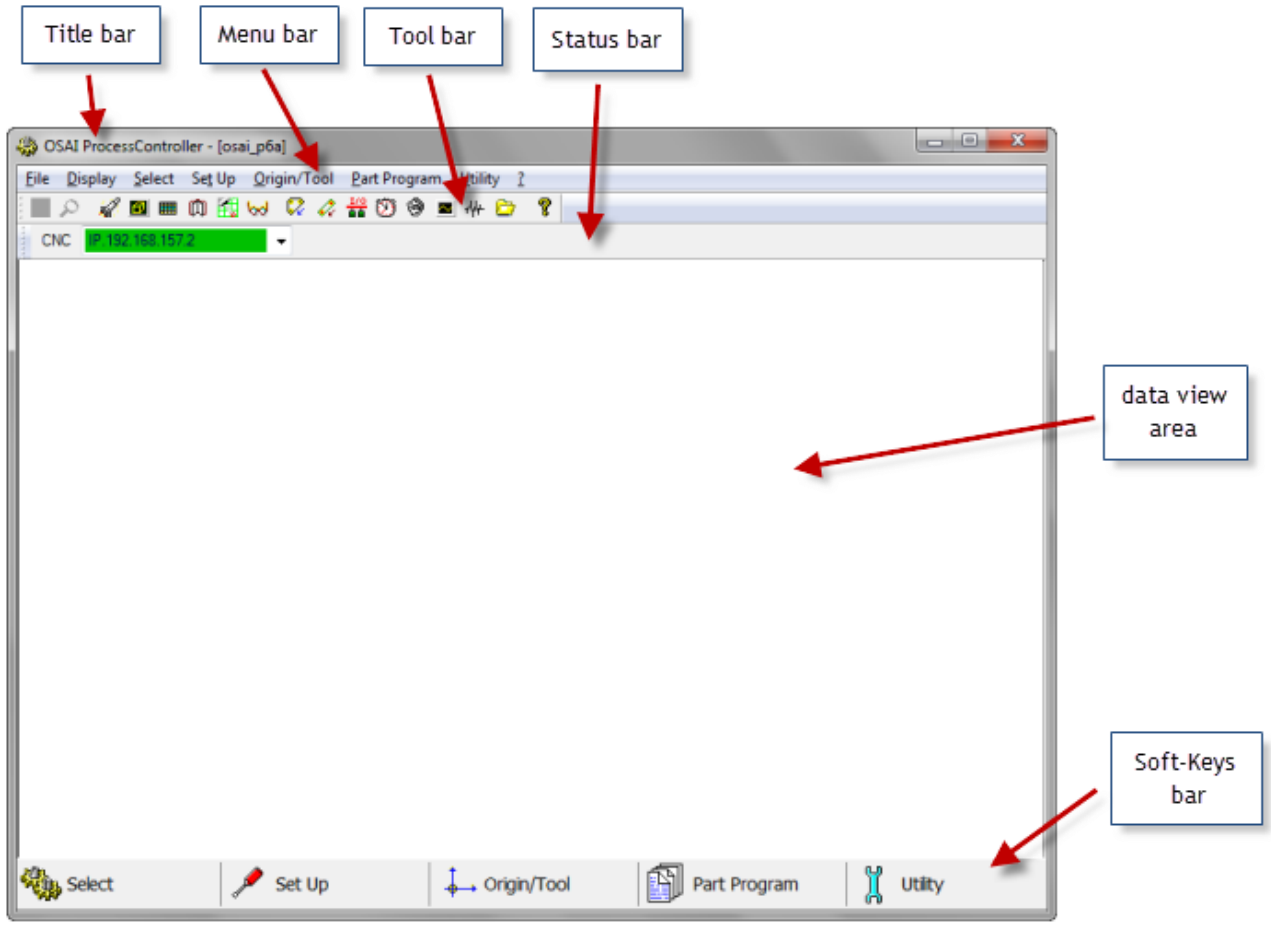
System History This is a standard feature that stores in a file all the diagnostic messages produced by the system in case of malfunction or incorrect operation, together with time and date. It is a useful tool for fault diagnosis.

1.4.1 Connection between the WinNBI and the CNC

As described in the previous paragraphs, the WinNBI applications can be installed on both the PC dedicated to the HMI and the control unit. In both cases, the applications connect to a data server in the control unit. The connection between the WinNBI Application and the NC, is always done via the network. The network will be virtual for the WinNBI locally installed on the CNC and real in case of the PC installation on the network. In all the WinNBI applications, the user can select the numerical control to which the application has to connect. The connection status is displayed in a dedicated status bar.

1.4.2 Common features of the WinNBI applications

Even if activated separately, the WinNBI applications are designed to offer the same graphic structure and common elements for ease of use. The basic graphic structure of a WinNBI application is as follows:



Title bar: Shows the name of the application and in some cases the name of the document (typically the file name) to which the application refers for the data display.

Menu Bar: Contains the main menu commands. These commands open a scroll-down menu listing a group of commands or a data insertion panel. The menu bar always contains all the entries in the Softkey bar together with other entries less used by the operator.

Tool Bar: Contains the buttons enabling the most used application functions. These can be direct or indirect commands. Every WinNBI application can have several or non-toolbars.

Status Bar: Displays the name of the Numerical Control to which the application is connected and the connection status via the virtual or real network.

Data view area: This is the area where the application displays the information and the graphic objects dedicated to the specific functions required by the application. Each WinNBI application has data and control elements of different types in this area.

Softkey bar: It is a bar that, in all applications, has 5 buttons that can be enabled using the mouse or touch Screen. For quicker activation, the buttons are associated with the function buttons F6 and F10 of the operator panel keyboard. These 5 buttons open the scroll-down menus containing the most used commands (see also Menu bar description).

In some application, the display of the field described can be customized. Their presence on the video depends on the settings made by the OEM and by the final user.

2. System Start-up

To turn the system on and off, comply with the machine manufacturer's instructions. To turn off the Numerical Control, follow the shut-down procedure described below.

2.1 System Power Up

The system is powered up when power is supplied to the central unit and the front panel, typically this is done by operating the machine's main power switch. Since most of the system software is stored on the hard disk, all the software that remains resident for the numerical control operation is automatically uploaded from the hard disk at system boot.

The BootController is a utility of the WinNBI graphic interface that checks and manages the CNC power up. Using this application, it is possible to choose the CNC operating mode and to check the messages generated by the CNC when powering up. The main working modes of the CNC are "*Normal*", "*Emergency*", "*Setup*" and "*Service*" are defined as "**stable**". Each mode has a specific function (for details see the paragraph "Switching the system on in special operation modes". The other modes, e.g., "HW boot" and "SW boot", are transitory and lead to one of the stable modes. The BootController utility displayed by the OPEN*console* Operator Panel, in the Windows ambient, can be manually enabled or automatically launched at power up. This behaviour depends on the choices made by the OEM when installing WinNBI.

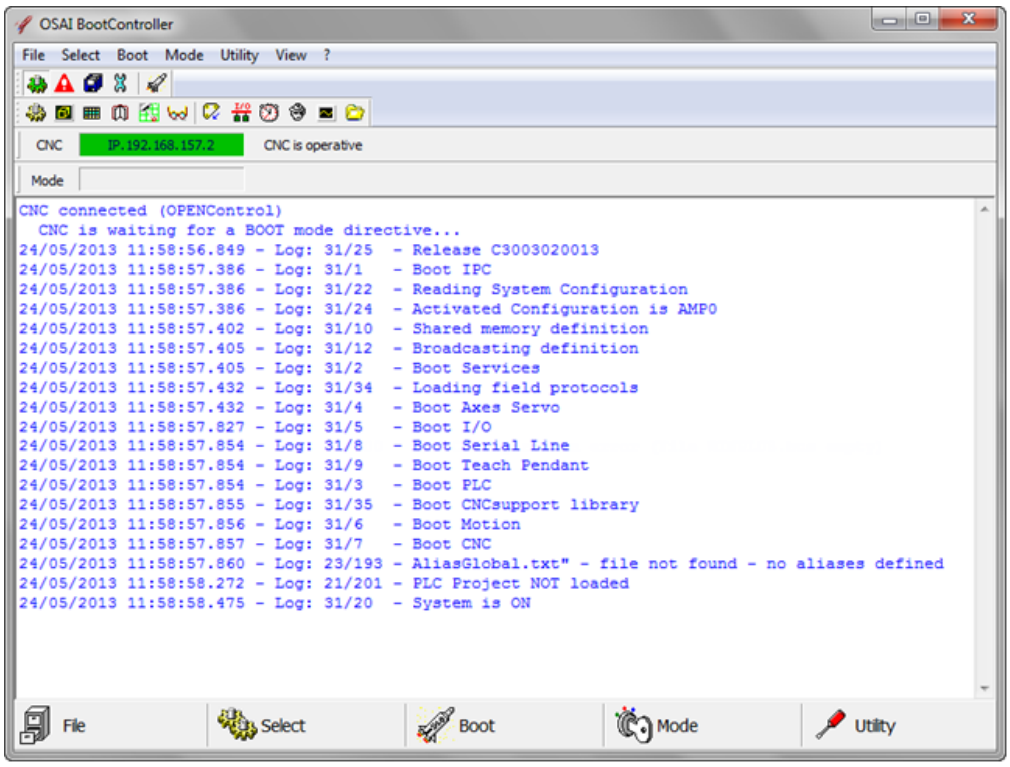
By default, the BootController, after connecting to the CNC and verifying that a stable mode has been established, automatically launches the ProcessController application for the system management.

2.2 System Shut Down

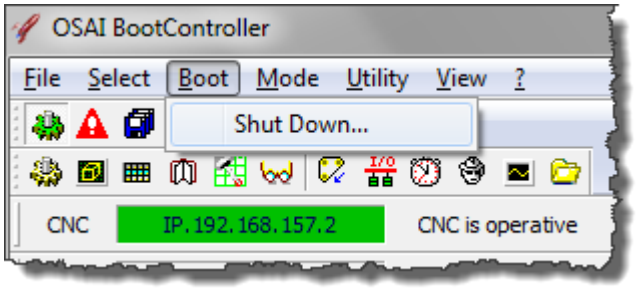
The CNC must always be shut down using the "**Controlled shut down**" procedure that closes all open sessions. This ensures the integrity of the data contained in the user, OEM or system files that, when shutting down, could still be open for writing.
Controlled shut down can be initiated during all working stable modes of the CNC. It is done using the BootController application.

The procedure is:

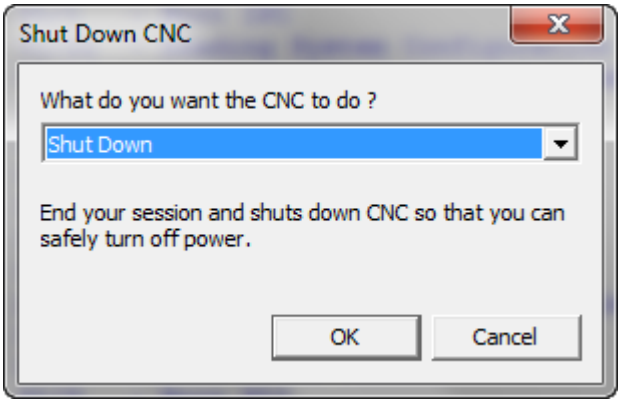
- **With the system on in one of the four stable operating modes:**
  If disabled, enable the BootController application


OSAI BootController
File Select Boot Mode Utility View ?
CNC
IP.192.168.157.2
CNC is operative
Mode
CNC connected (OPENControl)
CNC is waiting for a BOOT mode directive...
24/05/2013 11:58:56.849 - Log: 31/25 - Release C3003020013
24/05/2013 11:58:57.386 - Log: 31/1 - Boot IPC
24/05/2013 11:58:57.386 - Log: 31/22 - Reading System Configuration
24/05/2013 11:58:57.386 - Log: 31/24 - Activated Configuration is AMP0
24/05/2013 11:58:57.402 - Log: 31/10 - Shared memory definition
24/05/2013 11:58:57.405 - Log: 31/12 - Broadcasting definition
24/05/2013 11:58:57.405 - Log: 31/2 - Boot Services
24/05/2013 11:58:57.432 - Log: 31/34 - Loading field protocols
24/05/2013 11:58:57.432 - Log: 31/4 - Boot Axes Servo
24/05/2013 11:58:57.827 - Log: 31/5 - Boot I/O
24/05/2013 11:58:57.854 - Log: 31/8 - Boot Serial Line
24/05/2013 11:58:57.854 - Log: 31/9 - Boot Teach Pendant
24/05/2013 11:58:57.854 - Log: 31/3 - Boot PLC
24/05/2013 11:58:57.855 - Log: 31/35 - Boot CNCsupport library
24/05/2013 11:58:57.856 - Log: 31/6 - Boot Motion
24/05/2013 11:58:57.857 - Log: 31/7 - Boot CNC
24/05/2013 11:58:57.860 - Log: 23/193 - AliasGlobal.txt" - file not found - no aliases defined
24/05/2013 11:58:58.272 - Log: 21/201 - PLC Project NOT loaded
24/05/2013 11:58:58.475 - Log: 31/20 - System is ON
File
Select
Boot
Mode
Utility

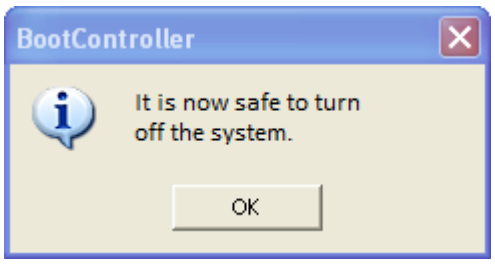
and select the "**Shut Down**" command from the "Boot" menu.



In the window, select the "*Shut Down*" command and confirm with the "*OK*" button


Shut Down CNC
What do you want the CNC to do ?
Shut Down
End your session and shuts down CNC so that you can safely turn off power.
OK
Cancel

when shut-down is completed, the following message appears, indicating that the CNC can be turned off.

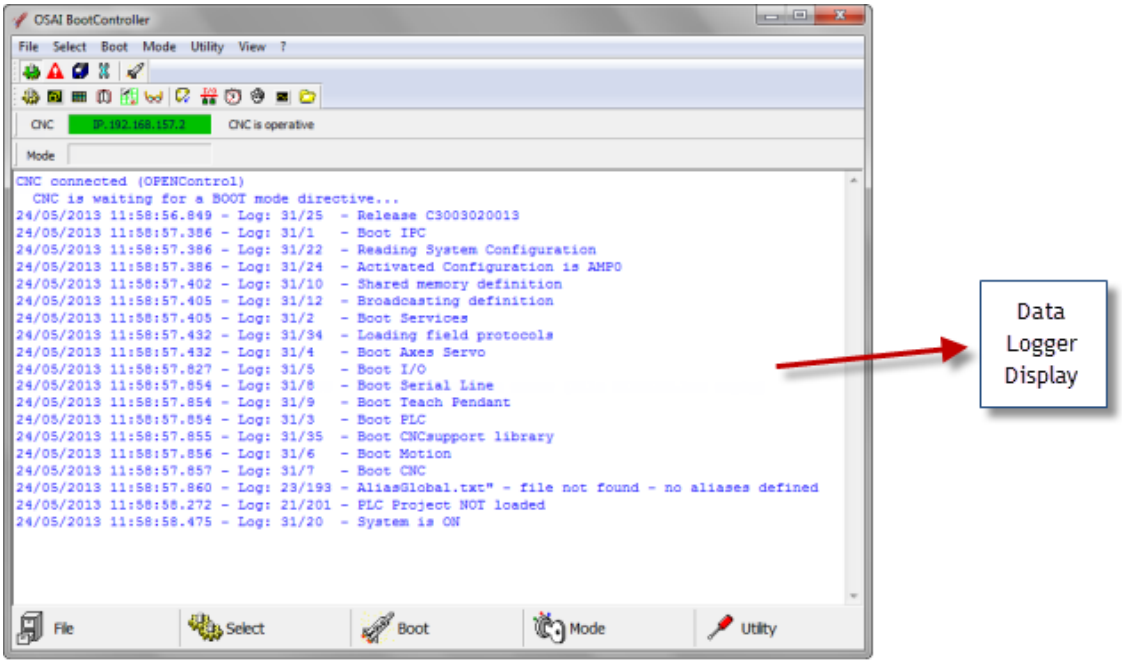


2.3 Diagnostic Screens

The control starts the diagnostic phase at boot up during which the hardware and software modules of the system are checked. The modules are tested in the following order:

- basic hardware (CPU, ROM, RAM, keyboard and disk)
- modules and additional devices
- numerical control software

The messages generated from the CNC diagnostics are displayed in the main window of the BootController application (*Data Logger Display*).
If the system is powering up, the messages are displayed as they are generated, but the same messages are recorded by the CNC and are visible in the Data Logger Display at any time, even when the boot phase of the CNC is finished.


OSAI BootController
File Select Boot Mode Utility View ?
CNC
IP: 192.168.157.2
CNC is operative
Mode
CNC connected (OPENControl)
CNC is waiting for a BOOT mode directive...
24/05/2013 11:58:56.849 - Log: 31/25 - Release C3003020013
24/05/2013 11:58:57.386 - Log: 31/1 - Boot IPC
24/05/2013 11:58:57.386 - Log: 31/22 - Reading System Configuration
24/05/2013 11:58:57.386 - Log: 31/24 - Activated Configuration is AMP0
24/05/2013 11:58:57.402 - Log: 31/10 - Shared memory definition
24/05/2013 11:58:57.405 - Log: 31/12 - Broadcasting definition
24/05/2013 11:58:57.405 - Log: 31/2 - Boot Services
24/05/2013 11:58:57.432 - Log: 31/34 - Loading field protocols
24/05/2013 11:58:57.432 - Log: 31/4 - Boot Axes Servo
24/05/2013 11:58:57.827 - Log: 31/5 - Boot I/O
24/05/2013 11:58:57.854 - Log: 31/8 - Boot Serial Line
24/05/2013 11:58:57.854 - Log: 31/9 - Boot Teach Pendant
24/05/2013 11:58:57.854 - Log: 31/3 - Boot PLC
24/05/2013 11:58:57.855 - Log: 31/35 - Boot CNCsupport library
24/05/2013 11:58:57.856 - Log: 31/6 - Boot Motion
24/05/2013 11:58:57.857 - Log: 31/7 - Boot CNC
24/05/2013 11:58:57.860 - Log: 23/193 - AliasGlobal.txt" - file not found - no aliases defined
24/05/2013 11:58:58.272 - Log: 21/201 - PLC Project NOT loaded
24/05/2013 11:58:58.475 - Log: 31/20 - System is ON
File
Select
Boot
Mode
Utility
Data Logger Display

The messages generated by the diagnostics are listed according to the test results and change according to the hardware and software system configuration, both in quantity and meaning.

Usually the messages are divided in several parts as the example below:

```
24/05/2013 11:58:57.432 - Log: 31/34  - Loading field protocols
```

From left to right, the message contains the following elements:

Date and Time Are the date and time of the message generation, with mill-second precision.

Ambient Indicates what part of the CNC generates it (e.g., “Log”).

Message Code Two numbers separated by a slash. The numbers identify the message (in this example "31/34"). The first number is the "*Class*" of the message, identifying a group or category of messages. The second number is the message "*Code*". Codes within a class are consecutive numbers starting from 1. The whole *Class+Code* group uniquely identifies a message or an error code of the OPEN*control*.

Message Text This column contains a brief description of the type of operation performed, the test passed by a device or the software section initialized. The message may be an error message if the operation has failed. The message in this example is "Loading field protocols."

***Appendix A** describes all the messages in detail and possible corrective actions.*

When the CNC detects a non-critical error, it indicates it through the message in the Data Logger and continues powering up the system. These errors are recorded in the **“SW0”** variable, accessible to the machine logic. The subsequent behaviour of the system depends on the machine logic defined by the OEM.
For additional details on the non-critical errors, see the documentation of the SW0 variable in the “OPEN*control* – Application Manual” code 45006962W.

2.4 Switching the System in Special Operating Modes

With the OPEN*control* system, it is possible to execute power on sequences different to the default.
These power up sequences put the CNC into special operating modes, activating only parts of the software system, allowing execution of such basic operations as:

- software installation or update
- calibration
- system configuration

Some of these modes are essential, for example, to remove user errors in the characterisation that prevents the CNC from starting properly.

The four OPEN*control* operating modes are:

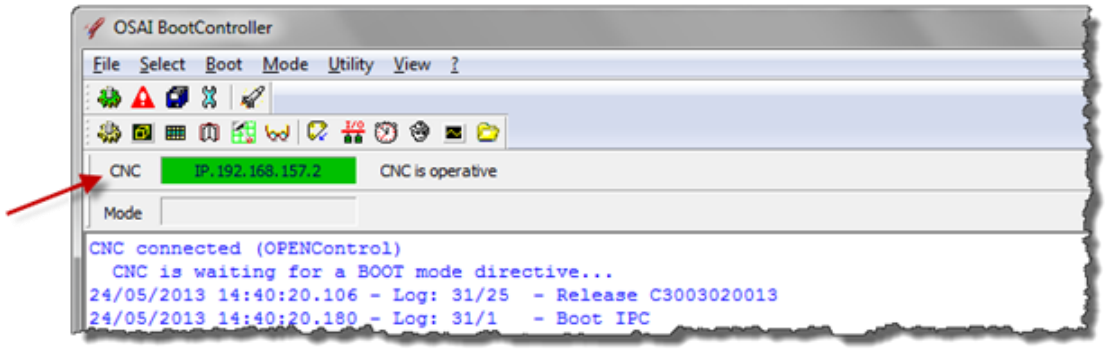
Normal Mode: it is the standard boot mode of the CNC, for the Part Program management and execution, axes movement etc.

Emergency Mode: the CNC works as a Server and only certain operations can be done, the main ones are:

- changing the AMP characterization using the ODM utility
- reset, save and restore the retentive memory via BootController
- reset of the CNC message history via the CNC System History utility
- CNC file management

Setup Mode: the CNC works as a File Server. In this mode, using the Security application, it is possible to upload the basic software, fixup or options to the CNC. It is also possible to save the CNC mass memory, to manage the security levels and option enabling (PAK) and to manage surface checks of the CNC disks, etc.

Service Mode: the CNC executes only the first boot stage of the CNC to initialize the axes control software. This mode is used for the calibration of the digital drives via the ODM utility.
The Emergency, Setup and Service modes can be set only via the BootController utility. If nothing is done with the BootController, the CNC will automatically select Normal mode.
All the WinNBI application indicate the operating mode of the CNC via a status bar that displays the name of the numerical control connected:


OSAI BootController
File Select Boot Mode Utility View ?
CNC
IP.192.168.157.2
CNC is operative
Mode
CNC connected (OPENControl)
CNC is waiting for a BOOT mode directive...
24/05/2013 14:40:20.106 - Log: 31/25 - Release C3003020013
24/05/2013 14:40:20.180 - Log: 31/1 - Boot IPC

The background of the field containing the CNC name changes according to the status of the connection between the WinNBI and the CNC. The colours used and their meanings are:

RED	CNC not connected. Connection in progress.
GREEN	CNC connected. CNC in “Normal” mode
ORANGE	CNC connected. CNC in “emergency” mode
BLUE	CNC connected. CNC in “Setup” mode
CYAN	CNC connected. CNC in “Service” mode

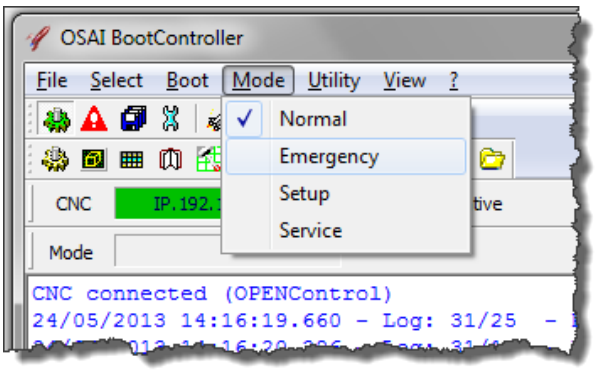
At the end of the CNC boot phase, in the working mode **Normal** or **Emergency**, if the option A03 is enabled on the system (see Application Security WinNBI), the BootController automatically starts the ProcessController application.

The ProcessController is the main application for numerical control management. The automatic start-up of ProcessController can be disabled via the "NBIconfig" configurator of the WinNBI.

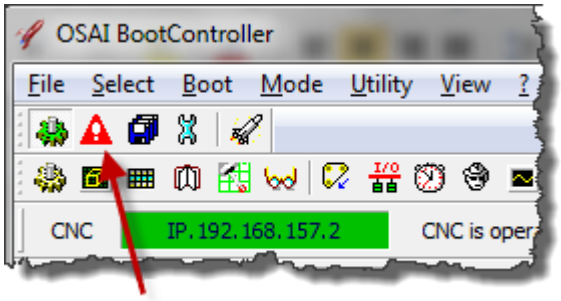
2.5 Selection of the Operating Mode

To select the boot-up operating mode of the CNC, the BootController is used. The procedure is the same for all the operating modes described in the previous paragraph.

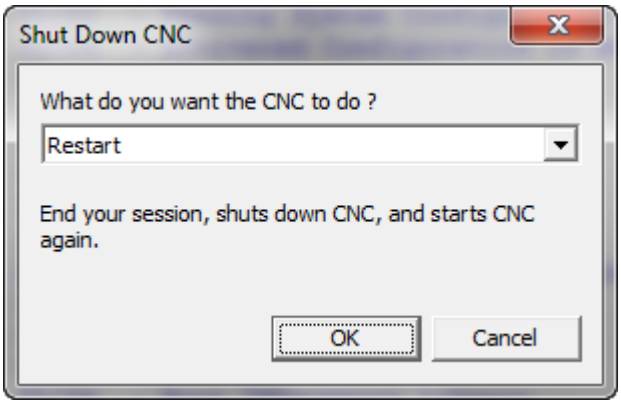
Selection of the operating mode with system on
To enable an operating mode, i.e., Emergency, select “Emergency” using the “Mode” menu



or with the dedicated button in the BootController toolbar.



After selection, reboot the CNC using the "*Shut Down*" command from the "*Boot*" menu entry.

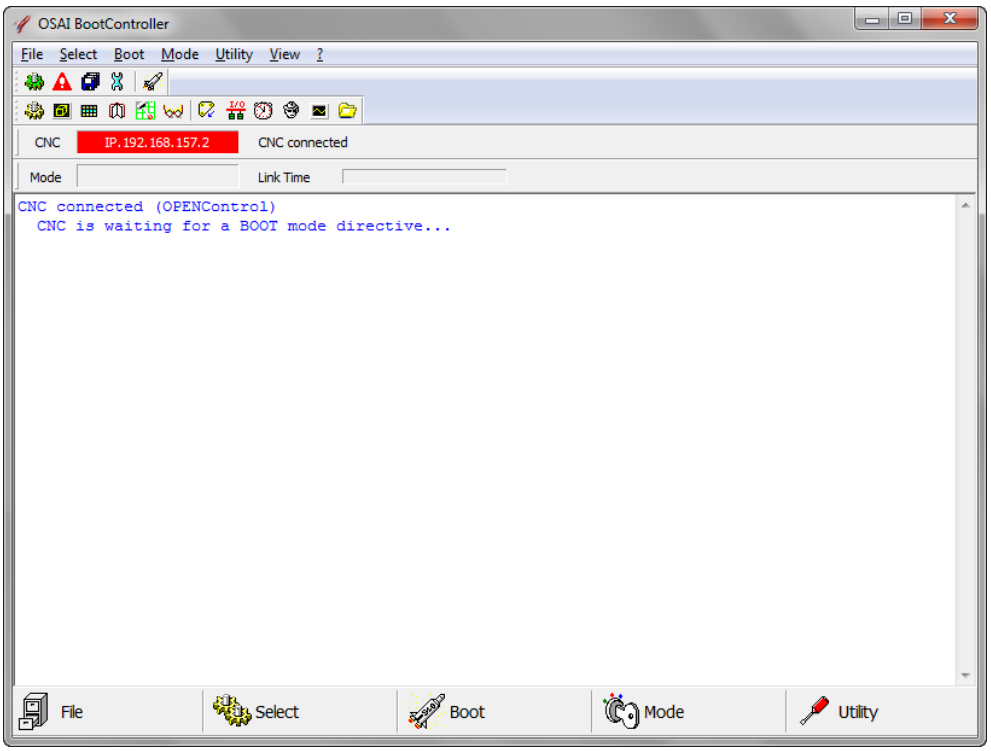


The system re-executes the boot phase, as if it had been turned off and on, and enters the EMERGENCY boot mode identified by the orange colour in the status bar.

Selection of the operating mode with the system off

Turn the system on. After the first hardware boot stage, the CNC waits for 30 seconds, for the selection of the operating mode from the Boot Controller. During this time it is possible, to select the desired mode using the buttons on the tool-bar. The CNC will continue the boot with the selected mode. If no selection is made, the CNC, continues to the Normal mode.

When the *BootController* is connected to a CNC waiting for the selection of the operating mode, the following page is displayed:



In emergency status the following WinNBI applications can be used:

- **SECURITY**
  Manages the security levels of the system for different user and installation levels. More information about this utility can be found later in this manual.

- **SYSTEM HISTORY**
  Utility for viewing / printing / deleting of the archived errors and system messages displayed during the machine operation.

- **FILEBROWSER**
  Management of the CNC's and the PCs OSAI file system.

- **ODM**
  Definition and change of the system configuration.

- **TABLE EDITOR**
  Management of the system's data tables (Tools, Offset, Origins and User). In Emergency mode, it is possible to work only on the tables file, not on the retentive memory files.

3. User Interface

The system displays data and information on the screen to start the communication, then the operator introduces commands and data. All these operations take place on the operator panel. The keyboard, video and the softkeys of the panels " OPEN*console* " and "OPEN*console* COMPACT" are described here.
The user interface is the means by which the operator communicates with the system.

3.1 The Screen

The screen is the means by which the system communicates with the operator, sharing all information concerning the functioning, the data associated with the operations carried out, the data entry requests, etc.
The screen can have a Touch functions to select field and graphic areas, to enable commands, etc.
With the OPEN*console* Operator Panel the information is displayed on a 17" or 19" colour screen with the Touch Screen option and 1280x1024 resolution. The COMPACT OPEN*console* Operator Panel is equipped with a 15'' monitor with 1280x1024 resolution and Touch Screen option.

3.2 The Keyboard

The OPEN*console* and OPEN*console* COMPACT Operator Panels have a 101-key U.S. keyboard.
The function buttons from F5 to F10 have a special function when the panel is used with the Prima Electro HMI software. The WinNBI, uses the function buttons for softkey activation. The F4 button is used for the selection of the video screen to display.

3.3 The Control Panel

The two Operator Panels have a section (Control Panel) used to send the operating instructions to the numerical control. The buttons provide the following functions:

The **RESET** button interrupts both the part program being executed and any active function.

The **CYCLE START** button initiates axes movements and other commands in both manual and automatic mode.

The **CYCLE STOP** button temporarily stops the active functions. For instance it stops the part program execution with a controlled deceleration of the axes. After a CYCLE STOP the operation can be restarted.

More detailed information on each button will be provided later in this manual.

The functions of the buttons described here those provided as standard by the system. However, they may be changed through the machine logic written by the machine tool builder. Refer to the MTB documentation.

3.4 The ProcessController

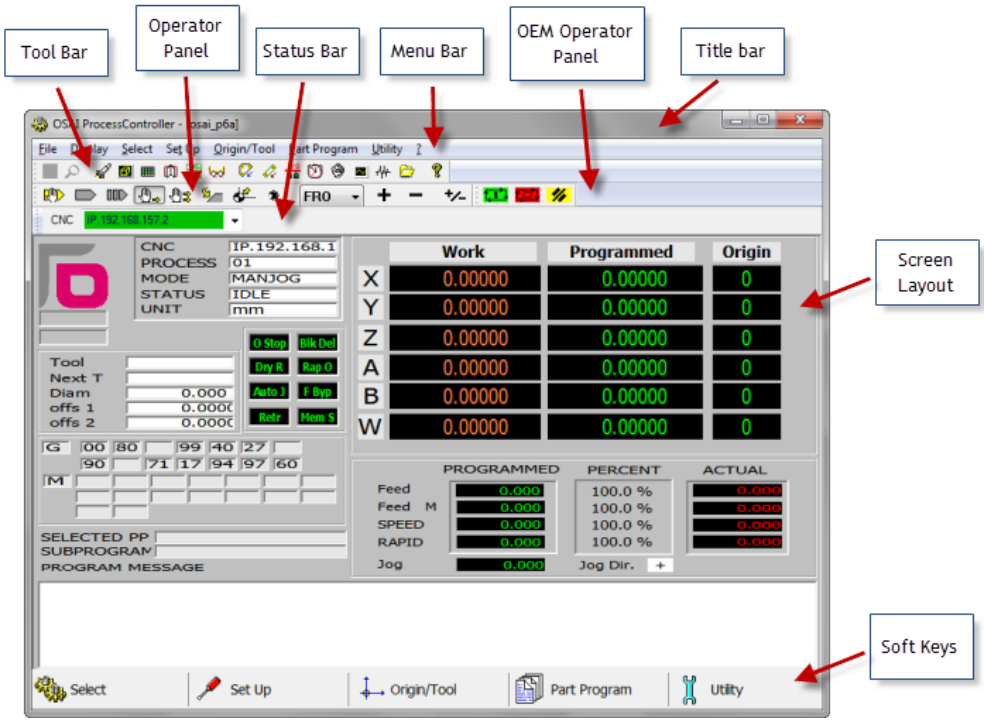
The ProcessController utility of the WinNBI displays and manages both axes movements and part programming of the CNC. For general features of the WinNBI applications, refer to Ch.1-“Features and specifications”.

For proper use of the ProcessController, it is important to understand the concept of “process”. “Process” (or “channel”) means the management of a machine tool, i.e., the control of the axes, the part program execution, the enabling and disabling of the machine logic control, etc.
More generally, process can also refer to a support activity for the machine tool, such as the tool change, management of part loading and unloading, calculations, statistics, reports, etc.
The term multiprocess represents one of the most important features of the OPEN*control* numerical controls. A single CNC can manage un to 24 different processes. The processes work simultaneously, each executing a part program independent of the others. If the systems controlled by the individual processes must be synchronized, they can be controlled more quickly and with greater flexibility than with separate control systems. The standard screens provided by Prima Electro with the ProcessController, display the data of only one process, so that they can be used on any machine. The standard screen can be used for multiprocess/mono-process systems. For the multiprocess systems, it is enough to select the desired process to ensure that the data on the screens are automatically updated with the selected process.

This manual describes the use of the standard screens provided by Prima Electro the software product release. The layout of the screens can be changed by the OEM and can then appear in a very different way to the end user. In this case the OEM describes the functionality of the customised screens.

Even if the content of the screen changes, the commands and the data-entry enabled with the data entry menu or with the softkeys remains unchanged.

The picture below identifies the main part of the ProcessController screen



3.4.1 Title bar

Shows the name of the application and the name of the selected screen. The title bar can be hidden or restored with the menu command **"*Display*"-"*Toolbar*"**.

3.4.2 Menu bar

Contains the main menu items, each item includes a set of commands displayed in a drop-down menu. The items in the drop-down menu typically open a panel for the introduction of commands and parameters. The most frequently used menu items are repeated in the Softkey task bar. The menu bar can be hidden with the menu command "***Display***" - "***Menu***". To re-display the menu, press the keys "***Ctrl***" and "**M**" on the keyboard.

3.4.3 Tool bar

Contains buttons to activate all the WinNBI applications:

The buttons enable the other WinNBI applications:

BootController

MachinePlot

TableEditor

SystemHistory

Digicad

PathView

ODM

Tool Calibration.

I/O config

TimeMonitor

Security

O-Scope

O-Scope.NET

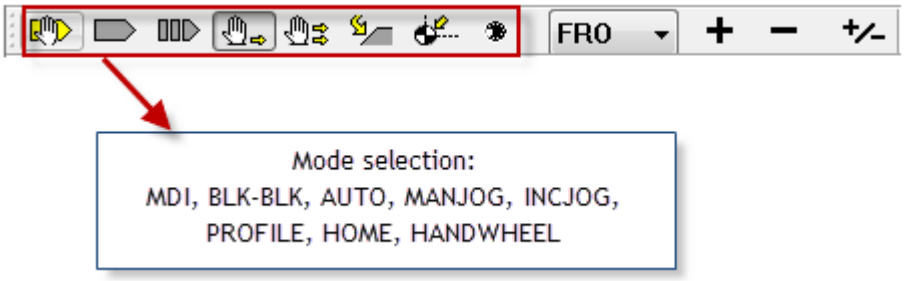
The button opens a panel that displays information referred to the software version of the ProcessController.

3.4.4 Operator Panel bar

It is a bar with buttons to select the CNC operating mode and the feedrate and spindle override values.

Mode selection

The CNC has different modes of operation, to move the axes in both manual and programmed mode. The operator panel allows the mode selection.



All the buttons select only the desired mode, without starting any machine movement. The machine movements must be confirmed with a dedicated command, usually the *Cycle Start* button.

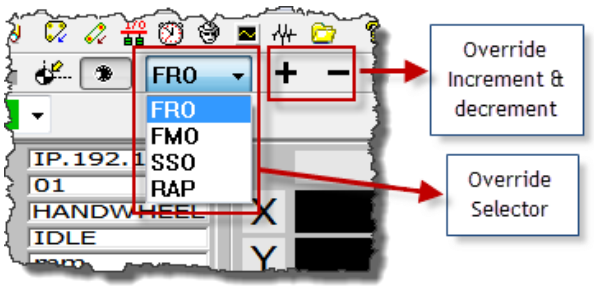
Code	Mode	Use
	MDI	Opens the MDI (Manual Data Input) data entry to allow keyboard entry of a program block.
	AUTO	Selects the continuous execution mode of the selected part program.
	BLK/BLK	Selects the block to block execution mode for the selected part program
	MANJOG	Enables continuous manual movements. Pressing the **[CYCLE START]** button, starts the movement of the selected axis, The movement stops when the Cycle Star button is released. For details, see Ch. 5 “Axes jogging and Stop functions”.
	JOG INCR	Enables incremental manual movements. Pressing the **[CYCLE START]** button, the incremental movement defined for the selected axis starts. For details, see Ch. 5 “Axes jogging and Stop functions”.
	PROFILE	Enables the return to the profile after a manual movement. For details, see Ch. 5 “Axes jogging and Stop functions”.
	HOME	Enables axes homing
	HANDWHEEL	Enables management of the handwheel

The selected mode is displayed on the screen in the “CNC status” area. See the description of the main video screen in following pages.

Selector and switches to change the override values

In the toolbar, are the override selector (percentage increase/decrease) and the buttons “+” and “-“ to change the value of the selected override.

Using the override selector it is possible to select the type of override that will be changed with the “+” and “-“ buttons.



The codes listed in the selector override values have following meanings.

Code	Definition	Use
FRO	Feed Rate Override	Percentage change in the cutting feed defined by Part Program with the "F" function.
FMO	Feed Manual Override	Percentage change in the feed defined in the AMP configuration for manual operations.
SSO	Spindle Speed Override	Percentage change in the rotational feed of the spindle programmed by Part Program with the "S" function.
RAP	RAPid override	Percentage change in the feed defined in the AMP configuration for rapid movements.

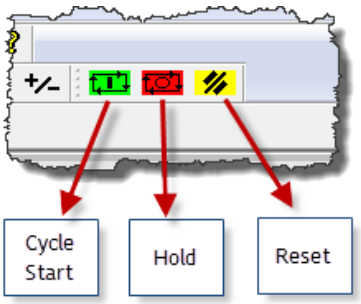
The current values of the override are displayed on the screen in the area “Feed and Overrides”. See description of the main video screen in the following pages.

Button “+”
Adjusts the feedrate, spindle speed or the rapid feed, depending on the function selected by the override selector. Each time the button is pressed, the percentage of the selected feed/speed is increased by a pre-set amount.

Button “-”
Adjusts the feedrate, spindle speed or the rapid feed, depending on the function selected by the override selector. Each time the button is pressed, the percentage of the selected feed/speed is decreased by a pre-set amount.

3.4.5 OEM Operator Panel

It is a password-protected button bar, for starting and stopping the movement. This bar is provided for the OEM only to allow initial machine tests which do not involve movement of the axes. The bar contains the Cycle Start, Hold and Reset commands.



3.4.6 Screen layouts

These may be the standard screen layouts provided by Prima Electro or one of the customised screens defined by the OEM.
Remember that the video screen structure can be changed by the OEM and therefore the screens displayed to the end user may be different to those described in this manual. Usually, the screens contain data of the process under control.
The ProcessController can automatically resize all the objects and the texts to adapt to the main window dimensions and therefore to the dimensions of the monitor used.

The table below lists the standard screens provided by Prima Electro with the numerical control software and the WinNBI.

File name	Screen description
osai_p6a.qv	Screen to manage program execution. Displays a maximum of 6 process axes.
osai_p9a.qv	Screen to manage the program execution Displays a maximum of 9 process axes.
osai_a5a.qv	Screen to manage the programs in automatic. Displays a maximum of 5 process axes.
osai_a9a.qv	Screen to manage the programs in automatic. Displays a maximum of 9 process axes.
osai_m9a.qv	Screen to manage up to 9 axes in manual mode
osai_aux.qv	Screen to manage the auxiliary axes of the machine logic

The default configuration of the ProcessController is set to display the sequence of screens *osai_p6a, osai_a5a, osai_m9a* and *osai_aux*.
The selection of a screen with the sequence above, is managed by the F4 key or via the menu command *"Select - Next page"*. The standard screen of the ProcessController dedicated to process control is described below. Remember that the screen setting depends on the OEM's choices and may vary significantly from those described in this manual.

This manual describes the "osai_p6a" screen containing all the display elements related to a process.

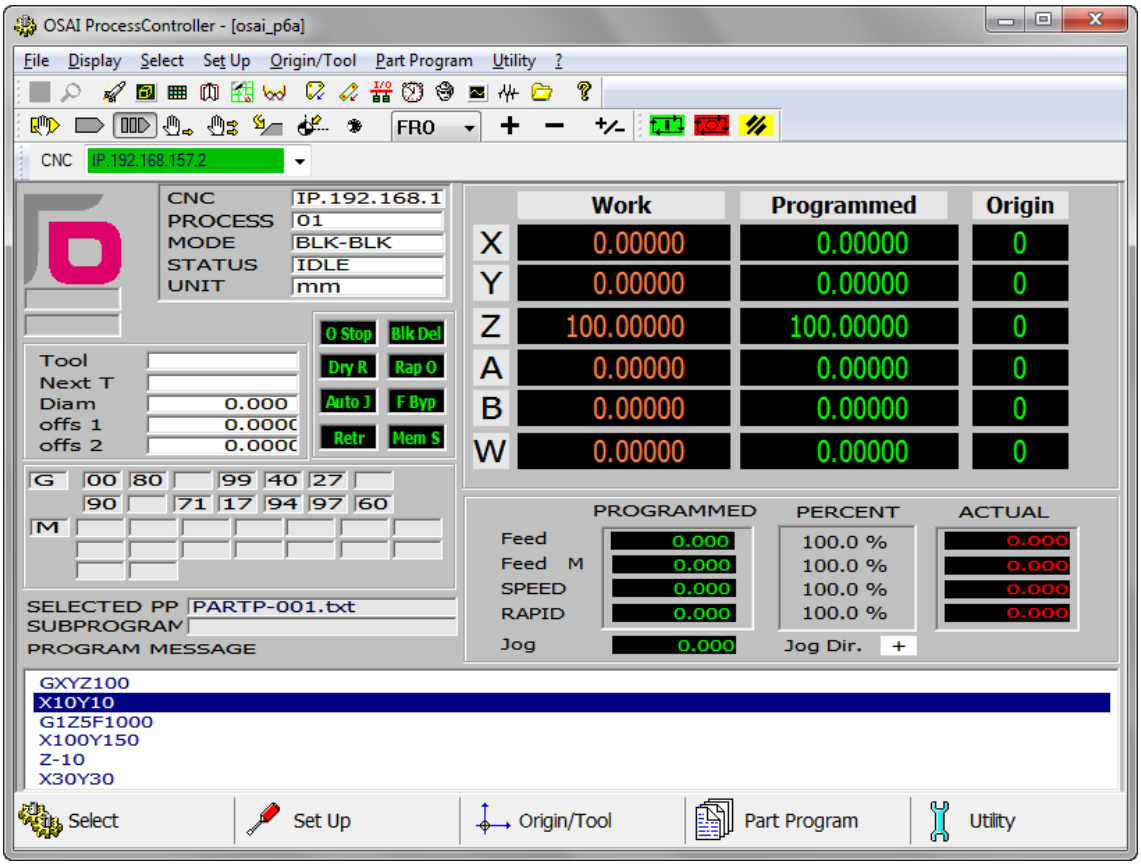
The other screens show the same data as in the "osai_p6a" screen, but rearranged to facilitate the use of the machine in manual or automatic mode.

3.4.7 SoftKey bar

Like the main menu items, each softkey selects a group of commands that are displayed in a drop down menu. The items in the drop-down menu usually open a panel for the introduction of parameters and commands. All the softkeys are repeated in the main menu.
The softkey bar can be hidden from and restored to view with the menu command "***Display***" – "***Softkey***". The softkeys can be activated, by using a mouse, touch screen or with the function keys F5 to F9.

3.4.8 Main video screen osai_p6a.qv

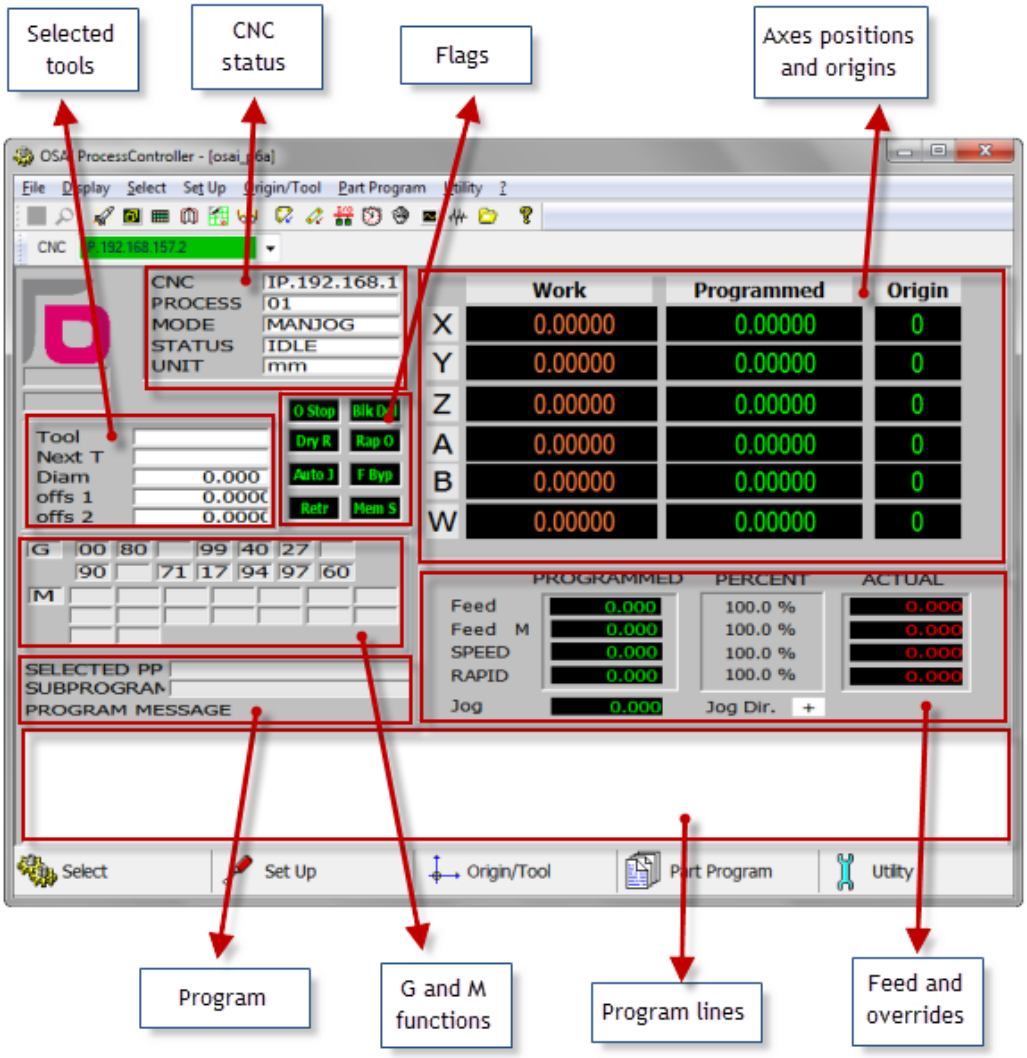
This manual describes the “osai_p6a” screen that contains all the display elements related to a process.



“osai_p6a” is a complete screen for the display of up to 6 process axes, that can be used for machining in all modes (automatic and manual).

The screen may be overlaid by other windows with additional information, these windows may appear automatically or on demand or according to user commands from menus and softkeys.

The ***osai_p6a*** screen has been divided into several functional areas, as shown in the following figure.



3.5 CNC Status Area

The Status Area contains information referring to the operating status of the numerical control:

Field	Description
CNC	Name of the Numerical Control connected on the network. The data displayed on the screen are referred to this NC.
PROCESS	Displays the number of the process selected and being controlled.
MODE	Displays the current operating mode, which can be either Automatic or Manual. The following belong to the modes: **Automatic**: AUTO, BLK/BLK, MDI **Manual**: JOG CONT, HOME, JOG INCR, JOG RETURN, HANDWHEEL
STATUS	Displays the status of the currently selected process. Some are valid for both Automatic and Manual modes, while others are possible in only one mode. The possible states are: **INPUT, ERROR, EMERG.**
UNIT	Indicates the current unit of measurement for the selected process. It can be **mm** (millimetres) or **inches**

Each of the aforementioned states is briefly described in the table. More detailed information will be provided in the explanations of operator actions in which they are involved.

STATUS	AUTO	MAN	MEANING
IDLE	●	●	The process is stopped waiting for a command.
IN CYCLE	●	●	The process is executing a part program or a command entered from the keyboard in MDI mode.
RUN		●	The process is performing a manual movement.
CYCLE STOP	●		The execution of a part program or a command provided in MDI was stopped and the process is in a wait state.
HOLD	●	●	The execution of a manual movement was interrupted with CYCLE STOP button and the process is in a wait state.
RUNH	●	●	The process has been stopped during the execution of a part program with **CYCLE STOP**. Switching the process in MANUAL, the status changes from STOP to HOLD CYCLE. If one or more axes are manually moved, during these movements the message becomes RUNH and buttons **CYCLE START** and **CYCLE STOP** are both turned on. RUNH informs the user that there is an on-going movement.
HRUN	●	●	With the process in the HOLD or CYCLE STOP state and **CYCLE STOP** is pressed, the status switches to HRUN which means that the process is ready to continue the interrupted machining**. The CYCLE STOP** button will turn off and the process waits for **CYCLE START** to be pressed**.**
RESET	●	●	Indicates that the process is still managing the reset command following pressing of the **RESET** button.
WAIT	●		The process is waiting to be synchronized with another process.
INPUT	●		Indicates that the process is waiting for a data input from the keyboard (a data entry window is on the screen).
ERROR	●	●	Indicates that the process has detected an error during operation. The error can stop the process or be managed by the part program.
EMERG	●	●	The system detected an anomaly, and operator intervention is required. The type of fault is indicated on the screen with an explanatory message explained in the Appendix A ("Emergency Messages").

3.6 Flags Area

The active options referred to the current mode, are displayed in reverse in the flag area

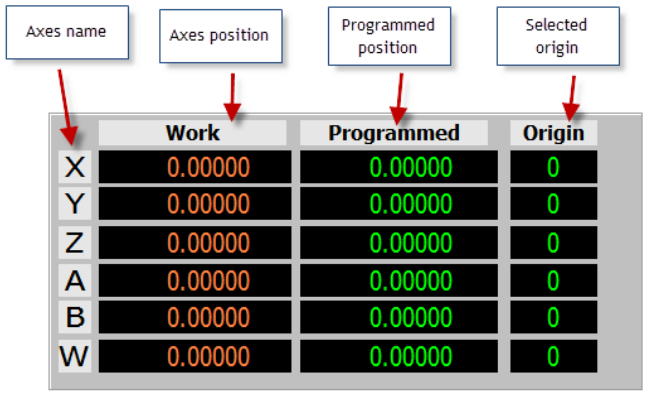
Flag	Description
OSTOP	Optional Stop (Ready to stop when M01 occurs in the program)
BLK DEL	Block Delete (blocks preceded by / are omitted
DRY R	Dry Run Program executed without axis movement
RAP O	Rapid Override
Auto J	
F Byp	Feed rate Bypass
Retr	Retrace
MemS	Search Memory

The table below lists (for Automatic and Manual mode) the sub-modes available, the possible process status and the flags used.

Mode	Sub-mode	Status	Flag
AUTO	BLK/BLK MDI	IDLE RUN HOLD HRUN RUNH WAIT INPUT EMERG ERROR RESET	OPTIONAL STOP BLOCK DELETE FEED BYPASS RETRACE RCM O N
MANUAL	HOME JOG INCR JOG RETURN HANDWHEEL	IDLE RUN HOLD HRUN RUNH EMERG ERROR RESET	RETRACE JOG+ JOG- RCM ON

3.6.1 Axes positions and origin area

This area of the main screen displays the position data the origins of the process axes. The figure below indicates the elements contained in the area. The data are arranged as a matrix where each line belongs to the axis whose name is shown in the first column.



Axes name

The axis name is a letter identifying an axis within a process. Unlike a physical axis identifier, that is a unique number in the system, the axis name can be repeated in another process.

The axis name in reverse indicates the axis selected for manual movements. The selection can be made in different ways:

- Clicking with the mouse on the axis name
- Sequentially, via the command menu or softkey "***Select***" – "***Next Axis***"
- By the Cursor Up/Down Keys
- Sequentially, by pressing "***Ctrl***" and "***A***" simultaneously.

Axes position

The label of this column indicates the type of positions displayed in the fields below. The content of this field changes according to the selection made by:

- the softkey or menu command "***Select***" – "***Positions***"
- Sequentially, from the keyboard by pressing the keys "***Ctrl***" and "***B***".

The position types that can be displayed for the process axes are:

Work position calculated with respect to the current origin
Machine actual position calculated with respect to the current origin
To go distance to go (difference between current and commanded position)
Error following error, i.e. the difference between the positions *Work* and *Machine*.
Absolute absolute position referred to the home position ignoring offsets & origins

The positions of the axes are displayed with 10 digits in the formats 7.3, 6.4 or 5.5 (where the first digit represents the integers, the second the decimals) according to what has been configured in AMP for each axis.

Programmed position

This is the last position programmed from part program or MDI command. It is 10-digits long, displayed in the same format as the "*Axis position*" fields.

Selected Origin

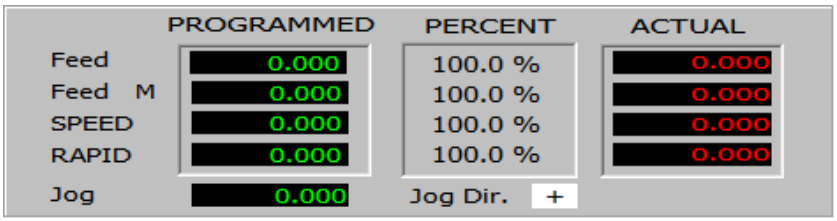
Number and type of the origin selected for the axis. In this field are also displayed characters:

- "T" for temporary origins
- "I" for incremental origins

If only the origin is displayed, then the origin is absolute, not temporary or incremental.

3.6.2 Feed and overrides Area

This area contains the information related to the machining speed and movement of the machine. The values in this area are expressed in current units of the process (mm / inches), which is also displayed on the screen.
The information in this area is arranged in a matrix where the lines refer to the type of speed indicated by the labels in the first column.



Feed line
Contains the information referred to the machining feedrate. From left to right, the data indicates:

- The feedrate set by program
- the selected feedrate override
- actual feedrate (programmed * feed override)

Feed M line
Contains the information referred to the manual feedrate. From left to right, the data indicates:

- the configured feed for manual movements
- the selected manual feedrate override
- the actual manual feedrate (configured * override)

SPEED line
Contains the information referred to the spindle speed. From left to right, the data indicates:

- the programmed revolutions. If the Constant Surface Speed (CSS) is active, the field indicates the calculated speed based on the programmed one which varies as the work diameter changes
- the selected speed override
- the override speed of the spindle (rpm or CSS) * override.

RAPID line
Contains the information referred to the axes rapid feedrate. From left to right, the data indicates:

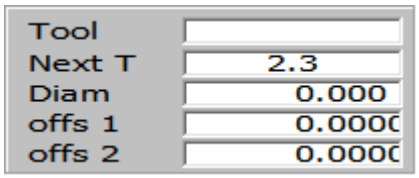
- rapid feedrate configured in AMP
- the selected rapid feedrate override
- the actual rapid rate (configured * override)

Jog field
It's the incremental distance set for the execution of manual movements (incremental Jog).

Jog Dir. field
Indicates the direction of movement set for manual movements (Jog and incremental Jog).

3.6.3 Selected Tools Area

The area is dedicated to the display of the programmed tools.



Tool

Number of the tool mounted on the spindle and its active offset. The field contains 2 groups of digits divided by a point. The group on the left of the point represents the tool number, the one on the right the offset.

Next T

Number of the next tool and its associated offset that will be mounted in the spindle. This is the tool already set by the program, but not yet mounted in the spindle.
The field contains 2 groups of digits divided by a point. The group on the left of the point represents the tool number, the one on the right the offset.

Diam

Diameter of the tool mounted in the spindle.

Offs 1

Length 1 offset of the tool. The machine axis to which the tool compensation refers to in this field is configured in AMP via the ODM application.

Offs 2

Length 2 offset of the tool. The machine axis to which the tool compensation refers to in this field is configured in AMP via the ODM application.

Offs 3

Length 3 offset of the tool. The machine axis to which the tool compensation refers to in this field is configured in AMP via the ODM application.

Offs 4

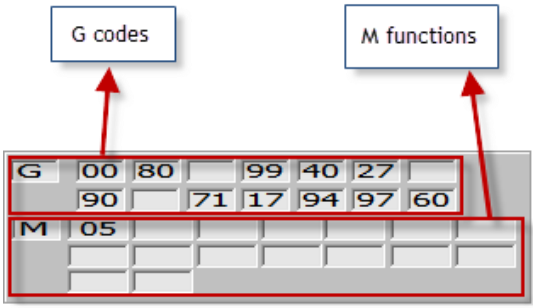
Length 4 offset of the tool. The machine axis to which the tool compensation refers to in this field is configured in AMP via the ODM application.

Offs 5

Length 5 offset of the tool. The machine axis to which the tool compensation refers to in this field is configured in AMP via the ODM application.

3.6.4 G and M functions Area

Area that displays the active G codes and M functions.



G codes

Active G codes. The codes are divided into display groups. Each field displays the active G code for one of the modal groups defined in AMP.

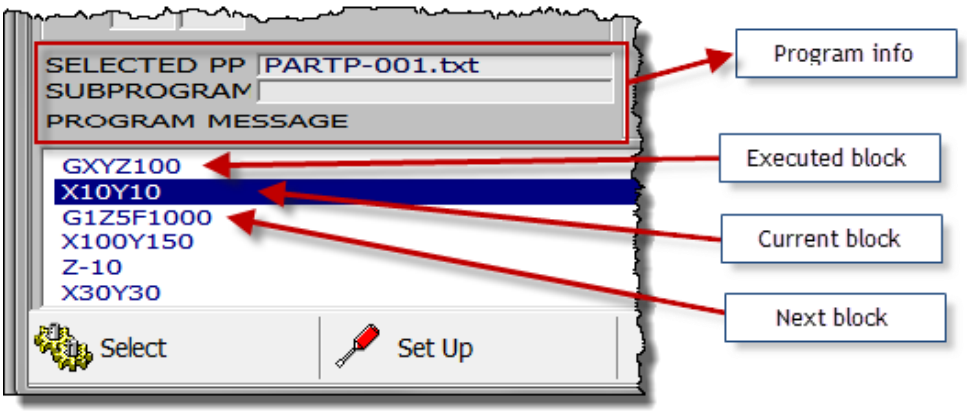
M functions

Active M functions. Usually each field displays an M code for every modal group configured in AMP.

3.6.5 Part program data area

This is the area dedicated to the display of the lines of the part program being executed. The area is read-only, at this point you cannot edit the program.

In order you will find, the block already executed, the block being in execution highlighted in reverse and the blocks still to be executed. The part program lines automatically scroll upward as they are executed by the CNC. The figure below shows an example of the lines of a program in execution:



SELECTED PP

It's the name of the main program selected.

SUBPROGRAM

Contains the name of the sub-program selected. The name is displayed only when the sub-program is enabled. A sub-program can call another sub-program. The sub-program displayed is always the one with the lower nesting level, i.e. the last called and still in execution.

PROGRAM MESSAGE

Messages defined in the part program with the DIS, command.

3.7 Keyboard Commands

The following tables summarizes the keyboard commands used as “accelerators” to enable the commands associated with the menu items.

Flag	Description
F4	Sequential selection of the screens to be displayed from a list of active screens
F5, F6, F7, F8 F9	Activation of the corresponding softkey
Ctrl + A, key ↑, key ↓	Sequential selection of the axis to be moved in manual mode
Ctrl + B	Sequential selection of the axes position type
Ctrl + P	Sequential selection of the selected process
Ctrl + T	Sequential selection of the selected target (CNC)

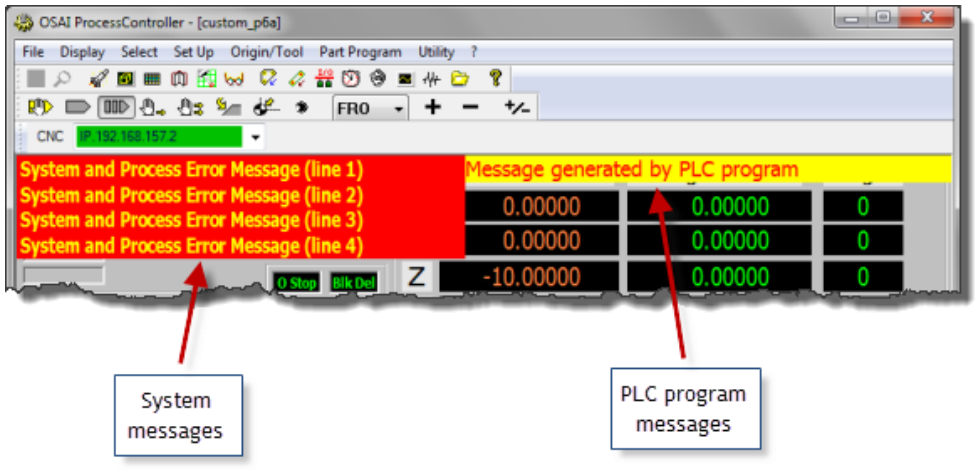
3.7.1 Additional information windows

In addition to the information normally provided by the main video screen, additional information in specific windows can be displayed as required. These windows overlay the current screen and appear only upon request or when the system has to inform the user (for instance, via a message).
The possible additional windows are:

MDI Command Panel
It is the panel for the MDI commands entry, i.e., for the execution of single command lines using the same syntax as used for the part program lines. For additional details, see the paragraph “***MDI Panel Functions*** ”

Messages from System
Information messages from the system or process displayed in four lines (see figure)

Messages from PLC Program
Signalling messages generated by the machine logic program. (see figure)

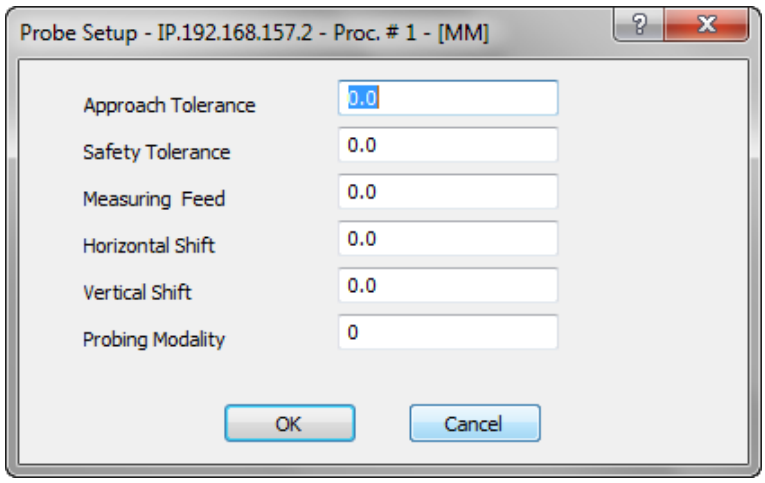


3.8 Panels for Command and data input

The ProcessController has several data entry windows used to enter commands, working parameters, configuration, parameter setting for the movements in manual and automatic. Usually the data entries are enabled by the commands associated with the softkeys described in the previous paragraphs.
This chapter explains the function of all the process Controller data entries.

3.8.1 General rules on the input windows

With the standard screens, the data can be entered in the system only by using specific windows defined as “input windows” or “data-entry”. usually, the data-entries appear in the current screen after enabling an item in the main menu or with the softkey.
For proper handling of input data it is necessary to apply some rules that are common to all input windows in the system. Below is an example of input window.

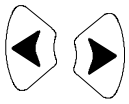


Each input window has fields that can contain parameters, file name, device names and so on. The values entered must comply with the type and the format of the field.

In the case of fields with parameters depending on the unit of measurement (mm / inches), the numerical values associated with them will be displayed in the unit of measure defined for the process with the default AMP configuration. The information is displayed in the title of the window.

The cursor is positioned on the selected input field. When the data-entry is opened the first input field is selected. With the operator panel touch-pad or with the touch-screen function, it is possible to select the desired field.

The characters inserted via the keyboard are displayed on the screen as the cursor is moved. Data entry can be managed using the following keys:

KEY	FUNCTION
	Moves the cursor to the right or to the left inside the field.
CANC or DEL	Deletes the character to the right of the cursor.
	Deletes the character to the left of the cursor.
	Toggles between overwrite and insert modes..
TAB	Moves the cursor on the following field
	Aborts the current data entry window.
	Confirms the current data entry window.

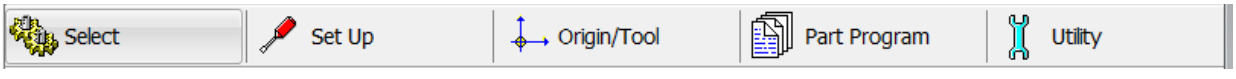
The input window can be exited at any time. If the data is not confirmed, the values inserted are not applied.
To exit the input window, cancelling the data entered, press "***Cancel***".
To exit the input window and validating the data entered, press "***OK***".

3.8.2 The ProcessController soft-keys

The ProcessController interface is controlled through the mouse or keyboard. To simplify access to all the CNC commands through the keyboard, some special keys have been added at the bottom of the application window, the soft-keys. The soft-keys are activated by pressing the OSAI Operator Panel F5, F6, F7, F8, and F9 function keys, the Touch-pad mouse or the Touch-screen function
The soft-key activation opens a drop-down menu. To select the desired menu item from the keyboard use the arrow keys (cursor up, cursor down).

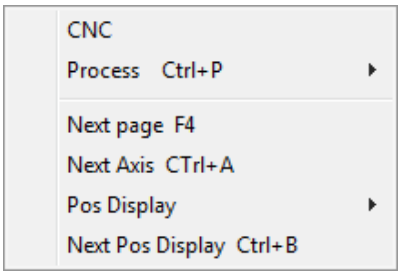
All the commands available with the soft-keys are also present in the main menu.

The ProcessController soft-keys are the following:

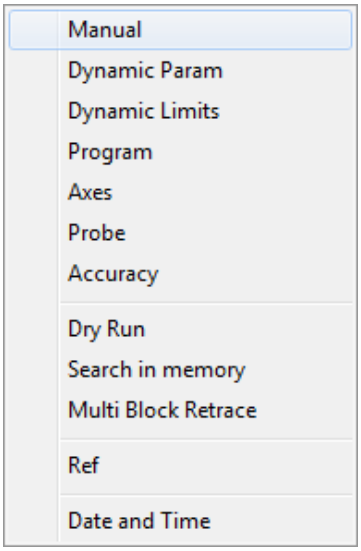


The drop-down menus opened by the softkeys are shown below:

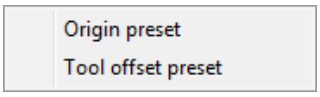
- ***Select***



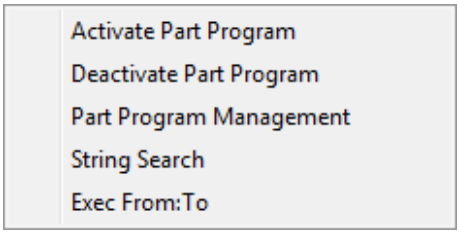
- ***Set Up***



- ***Origin/Tool***



- ***PartProgram***

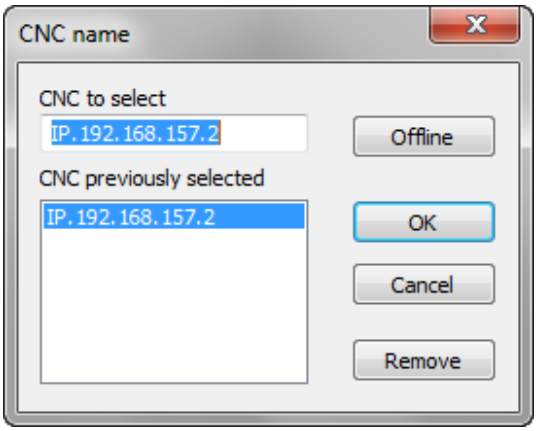


- ***Utility***

The panels opened by the drop-down menu items are explained In the following pages.

"Select" - "CNC"

This command changes depending on the configuration of the type of connection (single or multiple). The configuration is set using the configuration panel reserved for MTB use.
If the ProcessController has been configured for a single connection, the selection panel is as follows:

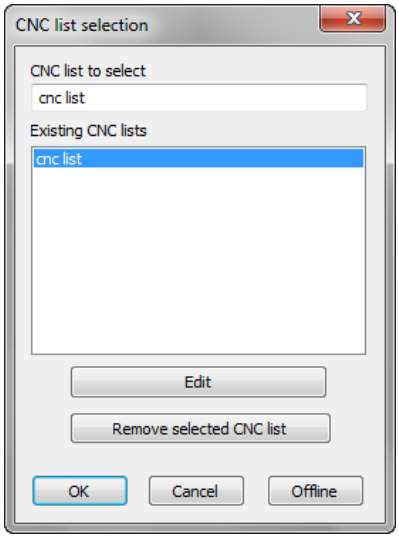


Permits selection of the CNC to be connected to the network.
When the WinNBI is connected to the CNC control unit, use the generic local address indicated by "IP." to address the local CNC. The remote CNCs are addressed by the prefix "***IP.***" followed by the TCP/IP address of the control unit (as shown in the picture). By using the ***Offline*** button or writing **NULL** instead of the CNC name it is possible to disconnect the CNC. This mode can be used by an offline PC in order to use the Layout Builder only.

When the offline mode is active the ProcessController displays an empty screen layout.
In this condition ProcessController doesn't try to connect to the selected CNC.
The "***Remove***" button is used to remove an unused name from the history of the CNC previously connected. The key removes the selected name from the "*CNC previously selected*" list.

3.8.3 Multiple connection, CNC selection

Multiple connection permits a list of CNCs to be specified to be connected contemporaneously.
It is used and useful only when the screen layout has been designed to show the data of more than one CNC at the same time.
If the ProcessController has been configured for multiple connections, the selection panel is as follows:

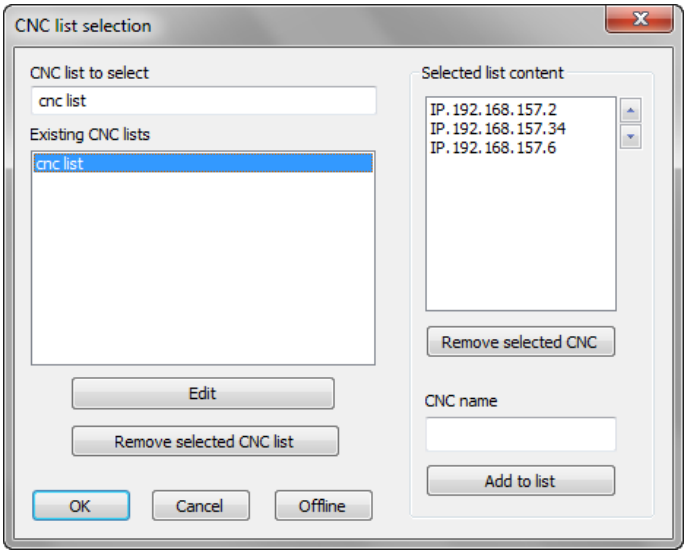


This panel is used to set the name of the selected list. The list contains the names of all the numerical controls to be connected simultaneously. To select the offline mode write “NULL” in the field “CNC list to select”.

The "Remove list" button may be used to delete an unused list (the button removes the selected name from the list).

To define the content of the list, press Edit.
The list selected is the one shown in the "CNC list to be selected" box.
If the list does not exist yet, a new empty list is created.

The Edit button opens (and closes upon being clicked again) the dialog window dedicated to the composition of the list. The dialog window can be edited as shown in the figure below:

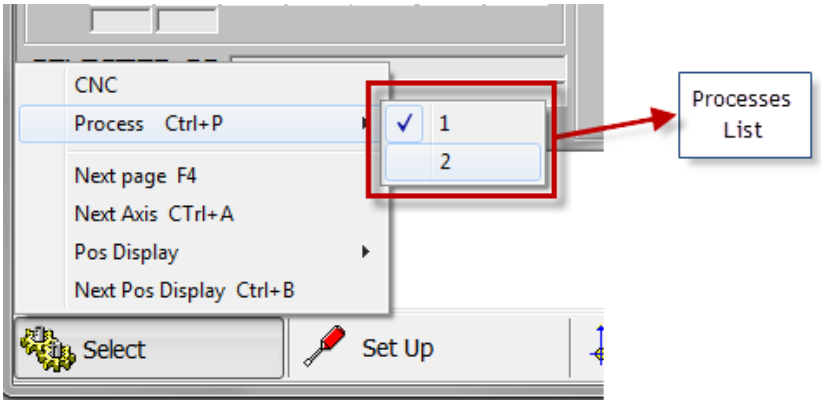


By means of the "Remove selected CNC" button you can remove the selected CNC from the list of CNC's to be connected. The "Add to list " button adds to the list the network name of the CNC specified in the "CNC name" field. The name is added to the list only if it not already present and if the list contains fewer than eight names.

The Up and Down arrow buttons next to the list of CNCs allows the position of the selected CNC name to be changed.

CAUTION:
The order of the names in the list is very important, since the information displayed in ProcessController screens depends on it. If a screen box has been configured to show a value of CNC # 1, the ProcessController will get the value from the first CNC in the list; if a box has been configured for CNC # 2, the value will be taken from the second CNC, and so on.
When a multiple selection is active, the ProcessController enables selection of one of the CNCs from the status bar, or sequentially using the keyboard command CTRL+T. The selection from the status bar is made by selecting one of the items in the list by means of the "Arrow down" button next to the "CNC" field.

"Select - Process"
Allows selection of a process from those configured in AMP.
The existing processes are listed in a drop-down menu.

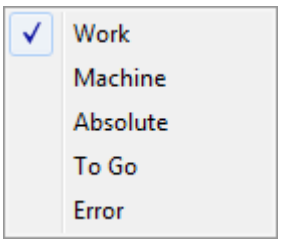


If there is only one process, the command is disabled.
If the screen is valid for the new process selected, the screen does not change, instead it will display the first screen of the sequence assigned to the process.
The command can also be sent from the keyboard by simultaneously pressing the keys "CTRL" and "P".

"Select" - "Next page"
Selects, sequentially, the screen following the sequence of the screens in use.
The command can be sent from the keyboard (key "F4").

"Select" - "Next axis"
Selects, sequentially, the axis for manual movements.
the command can also be sent from the keyboard by simultaneously pressing the "Ctrl" and "A".

"Select" - "Pos Display"
Allows selection of the type of position display for the axes configured in the process.

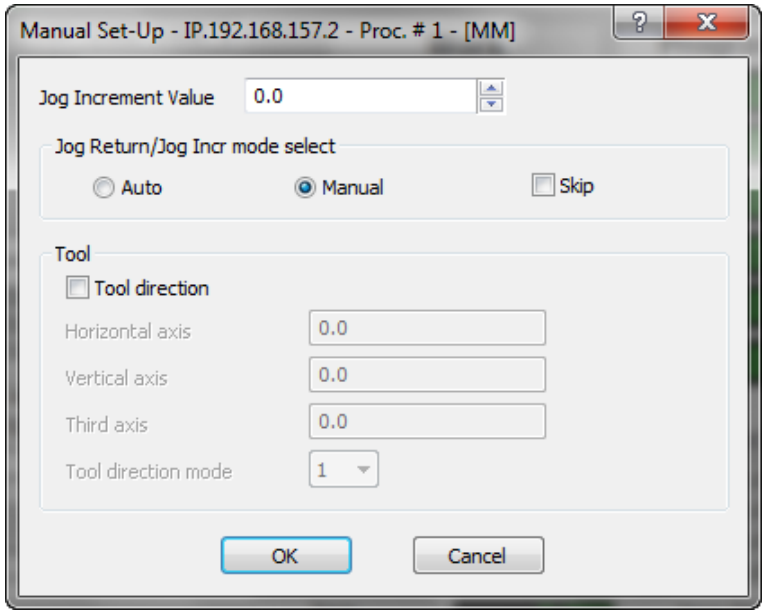


"Select" - "Next Pos Display"

Selects sequently the type of position display for the axes in the process. For each activation of the command the position type display can be selected in the order Work, Machine, Absolute, To Go, Error. The same command can be enabled from the keyboard by simultaneously pressing "***Ctrl***" and "***B***".

"Set Up" - "Manual"

Opens a data entry window that allows the length of the axis movement to move with incremental manual movments and the type of return after a manual movement. It also enables the tool extraction procedure for some particular cases (see Chapter 6, " Manual Axes Movements and Stop Functions ").



Jog Value

Insert the incremental distance for incremental manual movement. The value inserted is displayed in the JOG field of the main video screen.

Jog Return/Jog Incr mode Selection

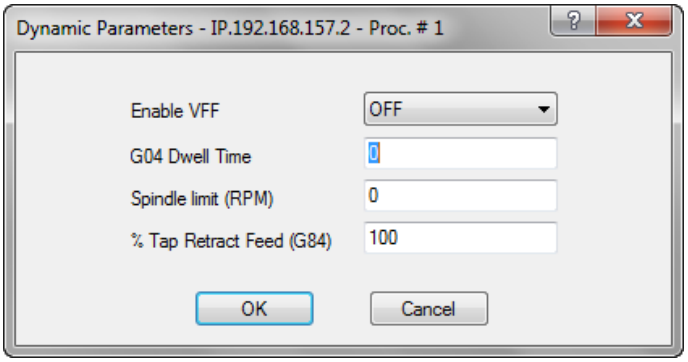
Specifies the JOG RETURN and the JOG INCR modes, that can be automatic or manual (See Chapter 4)

Skip

Specifies whether to return to the profile from HOLD, or to position along a straight line to the end point of the block in which the movement was suspended in HOLD. To skip a return on the profile, tick the Skip field.

For the fields related to the "Tool Direction", refer to Ch. 5 "Axes jogging and Stop functions".

"Set Up" - "Dynamic Param"



Enables VFF

Enables/disables the VFF algorithm *(velocity feed forvard)* of the servo axes.

When the algorithm is enabled, the following error will be reduced to a value close to *zero*, consistent with the gain of the servo configured in AMP, to minimize the shape error on the profile.

When the algorithm VFF is disabled, the axes are controlled only with the *following error*.

The value of the following error is displayed in the video area of the axis positions when selecting the "*ERROR*" display mode.

Dwell time G04

Defines the dwell time at the block end (G04). This time is used with G04 and in the following blocks of fixed cycles:

- In G94 the dwell time is in seconds.
- in G95 the dwell time is in number of revolutions.

For detailed information on this parameter, see the Programming Manual.

SPEED limit (RPM)

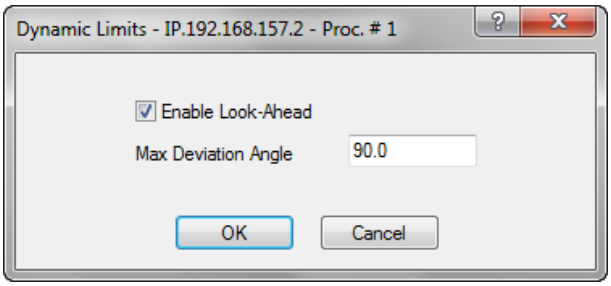
Defines the maximum spindle speed.

It is used in *Constant Surface Speed* (CSS), ie when the spindle feed depends on the *master* axis position.

Return feed % (G84)

Tapping return feed. It defines the percentage change in the feed rate applied during the return phase of the tapping cycle. The range of values is 0-125%. The default value is 100%.

"Set Up" - "Dynamic Limits"



Enables Look-Ahead

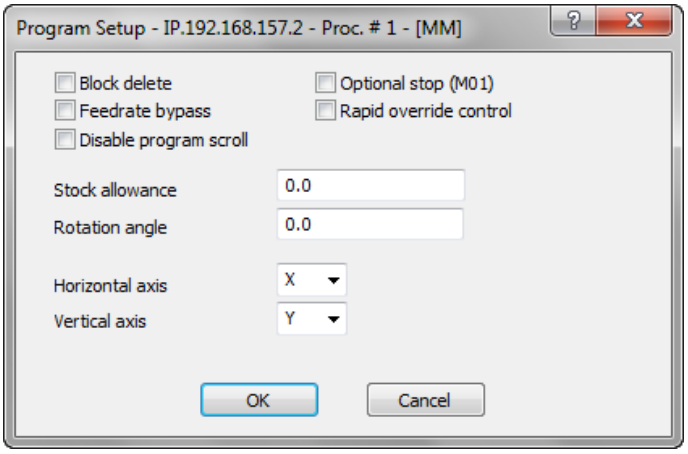
Enables the automatic deceleration on the angles when executing a part program. If disabled, all the DLA instructions of the current part program will be ignored.

Maximum Deviation Angle

MDA (Maximum Deviation Angle) is the maximum angular deviation from the axis in which G27 is active. The selected angle is the operational limit for G27. Angular deviation greater than this angle is done by G29 or point-by-point.

"Set Up" - "Program"

Opens a data entry that allows the configuration of some paraemters that influence the part programs execution modes.



Block Delete

Enables the block delete feature, the block preceded by a slash "/" is NOT executed (equal to the DSB = 1 programming). At system boot, this field is set to the condition established in the AMP characterization. System reset does not change the previous setting.

Optional Stop (M01)

Enables the optional program stop. At system boot, this field is set to the condition established in the AMP characterization. System reset does not change the previous setting.

Feedrate Bypass

It enables forcing of the rapid speed with G1, G2 or G3 active. At system boot, this field is disabled. System reset disables this field.

Rapid Override Control

Enables control of the rapid feed precentage with the Operator Panel. At system boot, this field is set to the condition established in the AMP characterization. System reset does not change the previous setting.

Disable program scroll

Disables part program scroll in the AUTO mode. If scroll is disabled, the block execution is faster. At system boot, this field is set to the condition established in the AMP characterization. System reset does not change the previous setting.

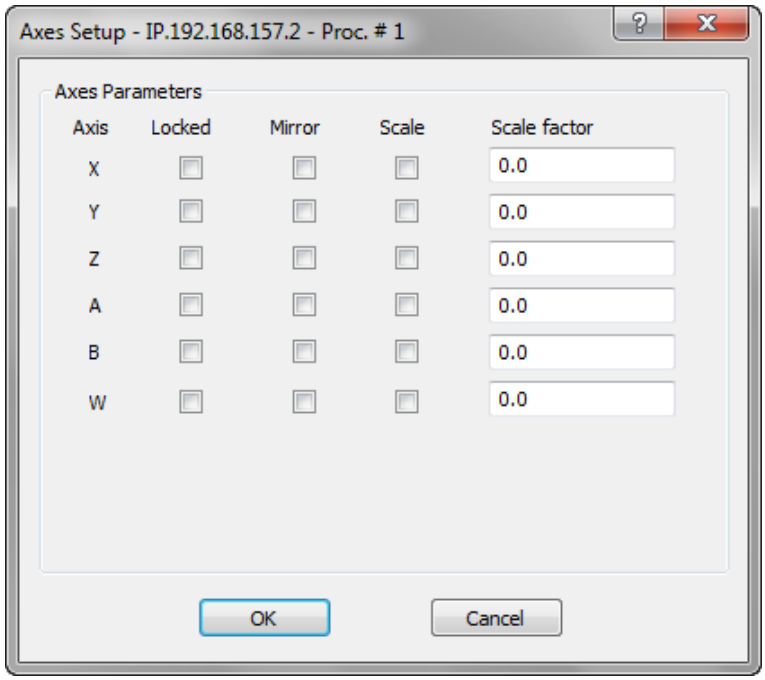
Stock allowance

Defines the machining allowance value added to the tool radius for the calculation of the trajectory in "tool offset" (G41 - G42) mode. It is the same as programming the MSA variable. At system boot and after a reset, this parameter =0.

Rotation Angle
Defines, in degrees, the rotation of the active plane (positive value = CCW rotation). Unlike the triliteral code by programming (ROT, angle), system reset does not reset the rotation angle. If (ROT angle) is programmed after setting up a rotation angle with this input box, the value of the rotation of the plane will be the sum of the rotations. Entering the angle of rotation of the plane in this window after programming a value with the trilateral (ROT angle), the rotation of the plane will be equal to the value entered in the data entry. In this case, the rotation preset by (ROT, angle) will be lost.

Horizontal axis – Vertical Axis
Defines the working plane. Ii is equivalent to programming G16. When the system starts-up and after a reset, the work plane is defined by the value set in AMP (G17, G18, G19 or the alternative axis plane (G16).

"Set Up" – "Axes Setup"



Locked
Enables the movement of the axes. When selected, the system normally execute the part program, but the axes do not move physically. The reset does not disable this mode.

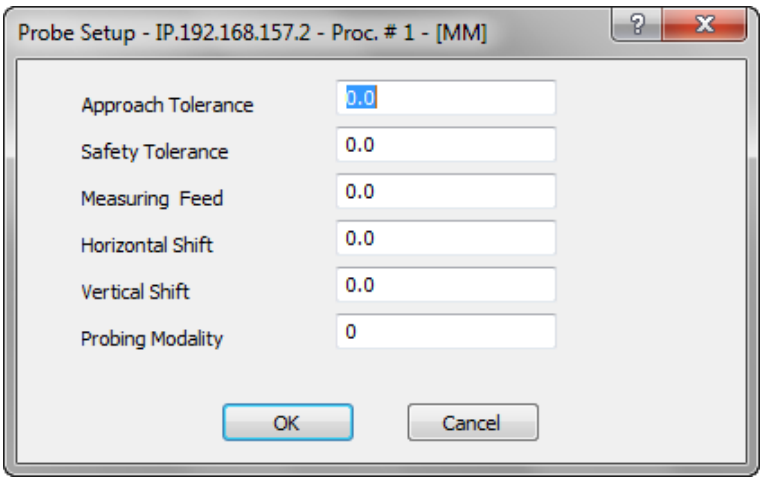
Mirror
Enable the reversal ("MIRROR") of the trajectory of the specified axis.It is equivalent to programming the three-letter code (MIR, axis name).System reset disables this function.

Scale
Enables the scaling factor referred to the axis specified. System reset disables scaling..

Scale Factor
Defines the scale factor to be applied to the specified axis. Equivalent to programming the triliteral code (SCF axis name, scale factor). System reset reset set the scale factor to zero.

"Setup" - "Probe Set Up"

Opens an input window for the entry of the probing parameters.



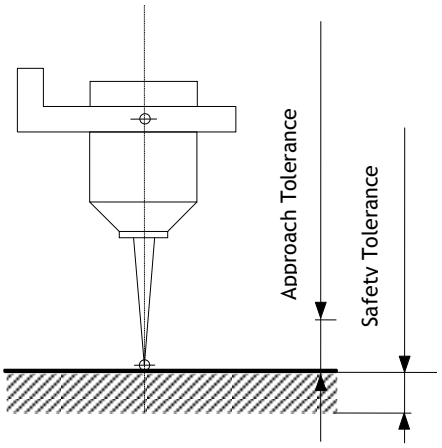
Approach tolerance. Represents the distance from the probing target. The probe moves this distance in rapid and then starts the probing cycle at the feedrate programmed in the entry **Measuring Feed.**

Safety Tolerance.

A value that prevents damage to the probe. If the probe does not find the target within this range, it stops.

Measuring Feed

The feed with the probe moves towards the target.



Horizontal Shift

The value for the abscissa axis of the probe requalification in millimetres.

Vertical Shift.

The value for the ordinate axis of the probe requalification in millimetres.

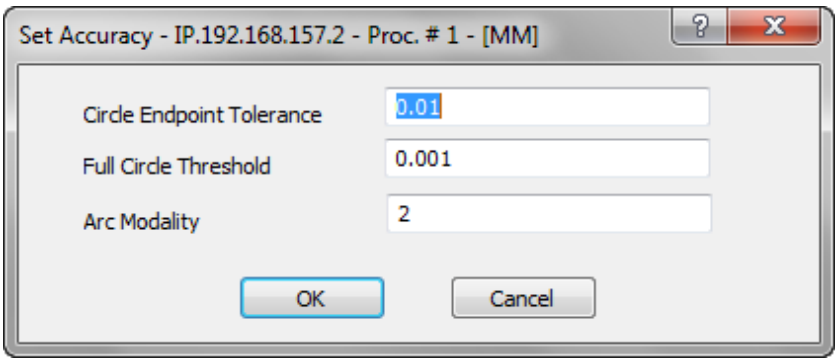
Probing Mode

The value of the probing mode:

- 0 for the standard probing (return to start point)
- 1 for the probe to remain at the probed position

"Setup" - "Set Accuracy"

Opens a data-entry to define the parameters related to circular interpolation.

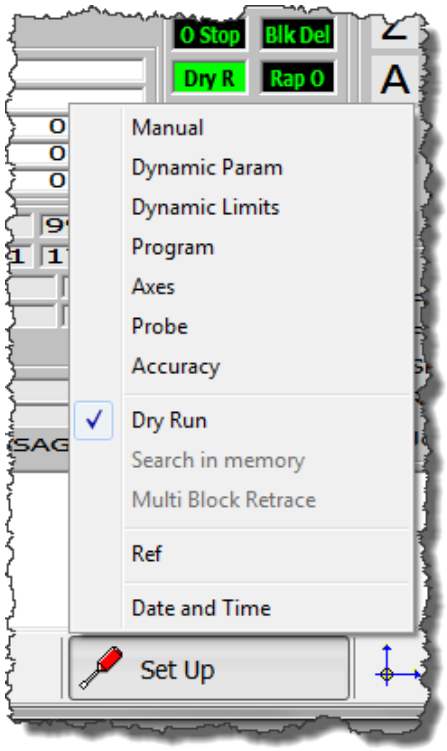


Circular Endpoint Tolerance (CET)
Defines the tolerance for the difference between the initial and final arc of a circle.

Full Circle Threshold (FCT)
Determines a tolerance for the distance between the first and the last point of the arc of the circle. If the distance is less than the value of FCT, the arc is to be considered a full circle.

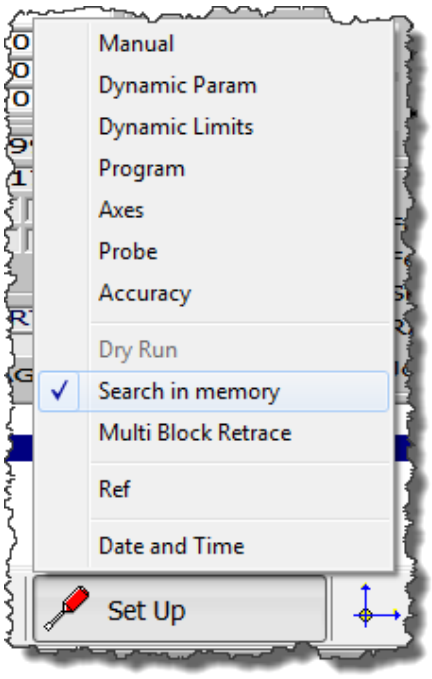
Arc Modality (ARM)
Determines the mode applied by the system to normalize a programmed arc with center coordinates (I, J) and the end point, in order to make it geometrically congruent. The valid values are 0, 1, 2 and 3.
0 displaces centre within the CET tolerance (default mode)
1 displaces starting point within the CET tolerance
2 displaces centre independent of the CET tolerance
3 displaces centre beyond the CET tolerance range

"Set Up" - "Dry Run"



If Dry Run is selected it is possible to test the part program execution without any physical axis movement or command being sent to the machine logic.

"Set Up" – "Search in Memory"

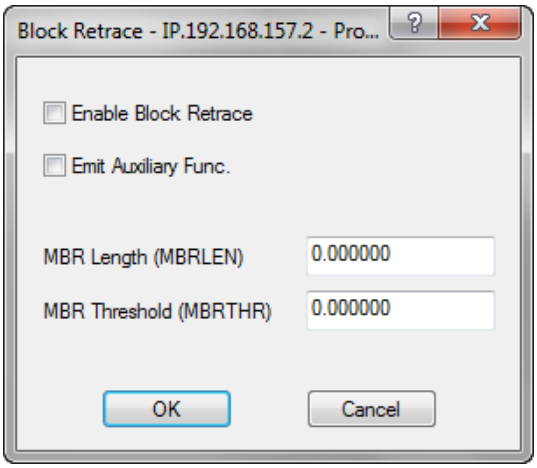


Search in memory: is the search for point in the program where it was interrupted and the subsequent recovery cycle or the search for a block where processing will begin. The parameters that uniquely identify a interrupt at run time are stored and continuously updated during the execution of the program. Based on these parameters, the control has the ability to perform an automatic search.

The strored search can be done in two ways:

- Stored search of the interruption block (following a shutdown or reset).
- Stored search of to a specific block

"Set Up" – "Block Retrace"



This function allows blocks to be executed backwards with axis movements in reverse along the profile previously executed. The number of blocks that can be executed backwards may vary from 1 to 64, depending on the AMP configuration.

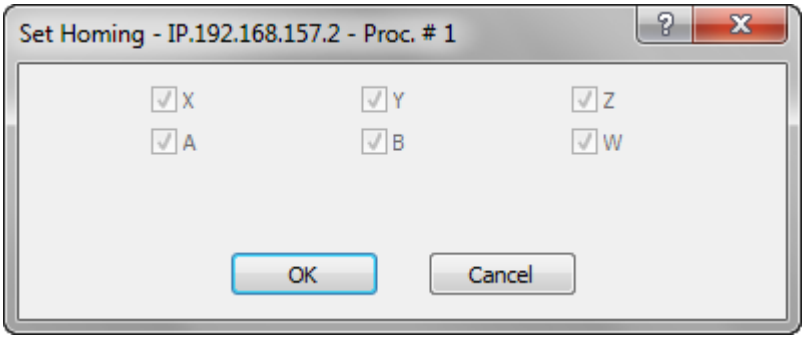
The sequence of the operations to execute the commands backwards is:

- Stop the running program by pressing **Cycle Stop**.
- Activate **Block Retrace** through the data-entry.
- Press the "*Cycle Start*" key: the system moves the axes backwards along the path stored during the program execution..
- The movements will be executed in selected mode (AUTO, BLK / BLK).
- Execution of Block Retrace can be interrupted by pressing the Cycle Stop command.
- To return to the standard conditions, disable **Block Retrace**, release **Cycle Stop** and press **Cycle Start**.

MBR Length
This field defines the maximum distance that can be run backwards. By pressing Cycle Start, the control runs backward, along the profile, for a distance equal to the value set.

MBR Threshold
Allows the definition of a distance threshold from an edge. If the reverse displacement moves the tool along the profile close to one edge, the control recalculates the end point along the profile to keep (before or after the corner) the minimum distance defined with this field.

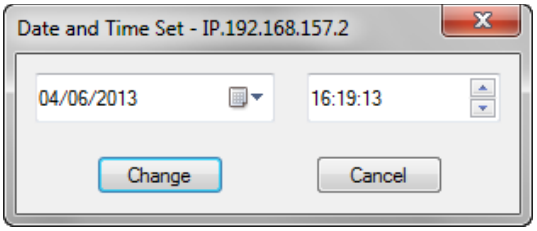
"Set Up" - "Ref"
Opens an input window allowing virtual referencing of the axes.



The box references the axes at the position where they are. Select the axes to refer and confirm by clicking "OK".

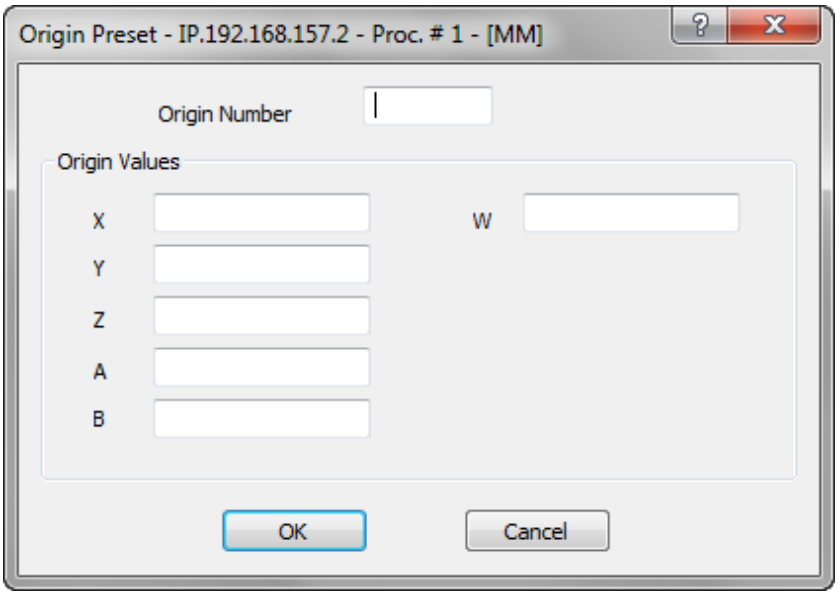
The axes referenced in this way, unlike the HOME cycle, is only a logical operation for which the parameters NULL OFFSET and HOME POSITION, configured in AMP for each axis, are not considered and the operating limits are referred to the point where the axes are upon confirmation.

"Date" and "Time"
Opens a box that allows the CNC date and time to be changed.



"Origin/Tool" - "Origin preset"

Opens an input window to define the origin values.



Origin Number

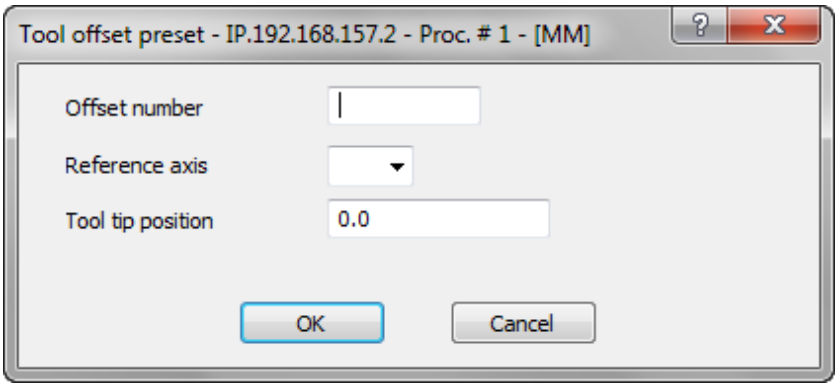
Number of the origin whose values are to be modified

Origin Values

Values of the axes to which the origin refers

"Origin/Tool" - Tool offset preset"

Opens a data-entry that allows the definition of an tool length offset



Offset Number

Offset number of the selected tool

Reference Axis

Name of the spindle axis (usually Z)

Tool Tip Position

Distance between the spindle "nose" and the fixed reference point

"Part Program" - "Activate Part Program"

Selects a part program for execution

"Part Program" - "Deactivate Part Program"

Disables the current part program.

"Part Program" – "Part Program management"

This command enables the management of the part-programs stored on the selected CNC. The management of the programs includes the following commands:

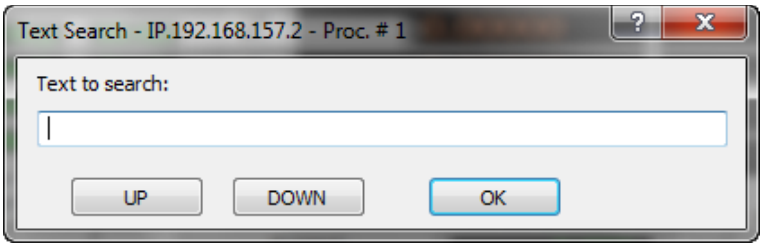
Selecting a program for execution.

- Deselect the active program
- Editing of a program.
- Changing the name of a program or a directory
- Erasing a program or a directory.
- Create a new directory or a new program.
- Copy / Paste of a program from a CNC directory to another.
- Copy / Paste of a program from PC to CNC and vice versa

For a full discussion of these commands see Ch. 6 "Part Program Management".

"Part Program" – "String Search"

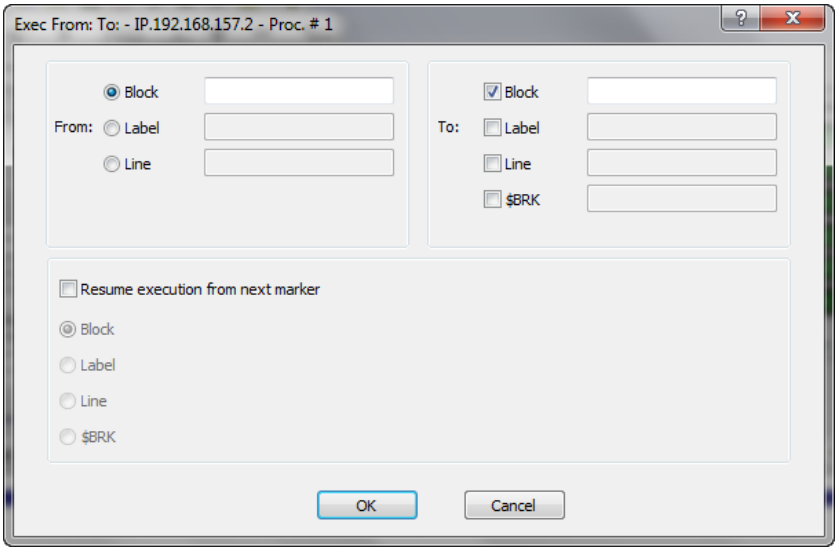
Searches the current program for a string of characters. The command is enabled only if a part-program has been selected. The input window appears when the command is initiated.



After inserting the text string; each press of the <Up> or <Down> buttons will search for the occurrence of a string in the desired direction. Press the <OK> button to terminate the operation.

"Part Program" – "Exec From To"

This command allows the definition of the part-program block (start and end) for partial execution of the selected program. The command is active only if a part program has been selected.



Allows setting of the partial execution modes of a part program; the number of parameters inserted determines the mode:

From	To	Mode
Specified	Specified	Executes the part program between the specified blocks
Specified	Absent	Executes the part program from the block specified to the end
Absent	Specified	Executes the part program from the current line to the specified block
Absent	Absent	Executes from the current line to the end

The starting block is defined using only one of the possible selections defined at point 3.
The conditions, described at point 4, determining the end of the program execution, can be more than one at the same time. The field $BRK identifies the equivalent variable used in the part programs, for this field indicate the value that the variable must have in order to finish the program.

***"Utility"* - Application name**
The "Utility" soft-key opens a drop-down menu that allows thw launch of the other WinNBI applications listed below.

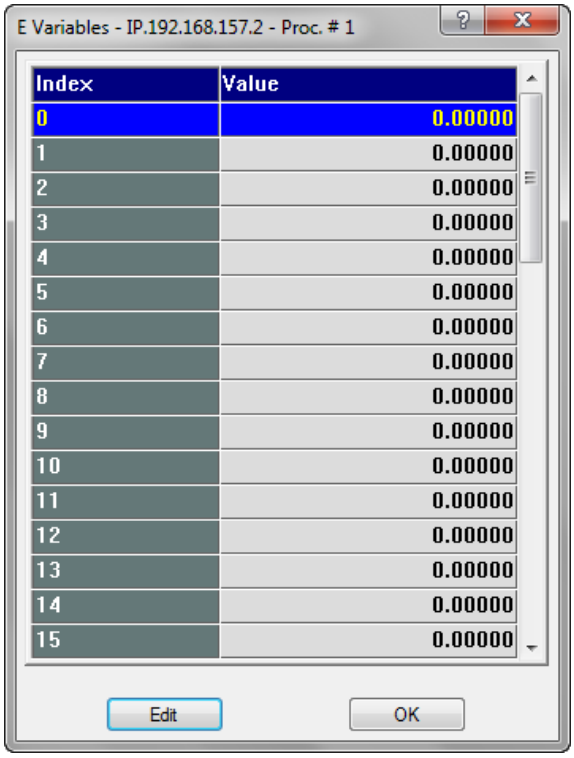
BootController, Machine Plot, TableEditor, SystemHistory, Digicad, PathView, ODM, Calibration Tool, TimeMonitor, Security, O-Scope, O-Scope..NET, FileBrovser

The OEM can disable these applications using the NBIconfig application.

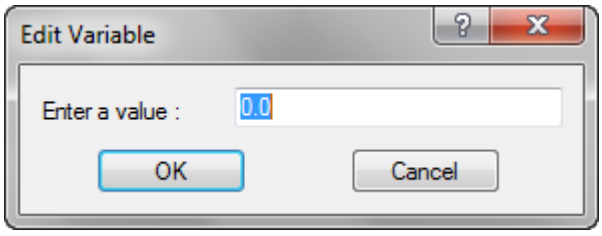
Utility - Variables
The "Utility" soft-key also allows the opening of a data-entry for the display and change of program and system variables.
The variables are divided by class and may be used for part-program writing, to obtain data from the system and from the machine logic developed by the OEM. The meaning of their content is therefore bound by the application developed for machine tool in use.
For viewing and editing of the parameters through the windows of the standard ProcessController, the variables can be divided into two classes:

- numerical variables
- alphanumeric variables (texts)

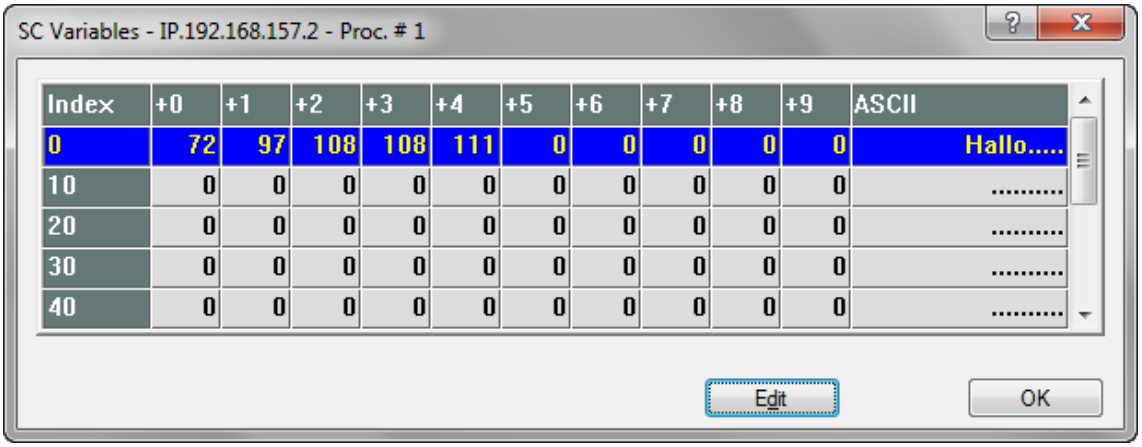
For a numerical variable, the Process Controller displays a panel like the one below:



The variables displayed are shown in a preset or configured length array.
To select the desired variable use the vertical scroll bar or the vertical direction keys (arrow up and down) on the keyboard. These operations move the display window through the array and allows the desired variable display to be selected.
To change the value of the selected variable, press "**Edit**", and the data entry appears, change the value and confirm with the "**OK**" button.

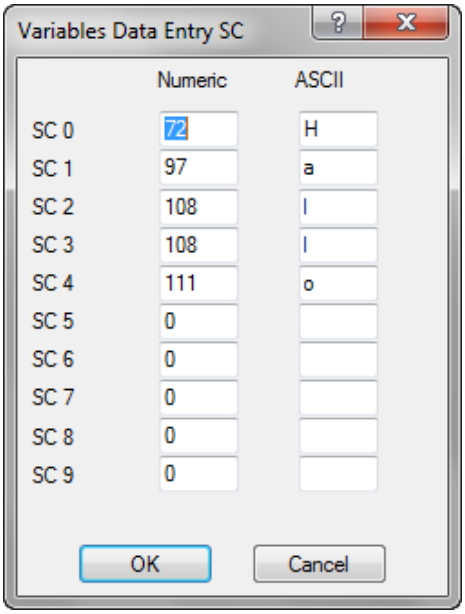


Selecting an alphanumeric variable, the Process Controller displays a panel like the one below:



The alphanumeric variables are displayed in the panel with both the numerical value and the text.

To select the desired variable,use the vertical scroll bar or the arrow keys as for numeric variables.
To change the value of the selected variable, press "**Edit**", and in the box that appears, change the value and confirm with the "**OK**" button.



The change can be done via numerical values (using the ***"Numeric"*** column) or by characters, inserting the characters in the ***"ASCII"*** column.

Using the data-entry of the *ProcessController* dedicated to the variables, it is possible to view and if allowed, change the variables E, SN, SC, User, H, Plus, Input and Output.
For additional details on the use of the variables and their format, refer to the Programming Manual.

3.8.4 MDI Panel functions

The MDI panel is used to set the command from keyboard that will be excuted by the CNC in MDI mode. The panel automatically appears when the "MDI" working mode is selected. Using the configuration panel, it is possible to choose how the MDI panel appears either as a basic data entry or it can be used to store the entered commands in order to re-execute them as a part program.
There are two types of panel.

Basic MDI panel:
The "Confirm" button sends the command entered in the command editing window to the CNC.
The "Change Block" button brings the current block of the selected part-program into the command editing window.The part-program block selection moves on to the next block.

Extended MDI panel:
When this panel is enabled, at each block "Confirm", the command is stored in a local file (specific for the CNC and the selected Process). Using the controls in the panel, the list of commands can be run, edited, deleted and saved to a file of the CNC.

The "**Start recording**" button deletes the current storage file (of the CNC and the selected Process) enabling the storage of the MDI commands. When the storage is enabled, an image representing a camera is shown next to the "Confirm" button.

The button "**Continue recording**" enables the storage of the MDI commands without erasing the previous ones. The new commands are appended after the existing ones. Note that the commands are stored in a file and therefore can be inherited from previous work sessions.

The button "**End recording**" stops the recording of commands and enables the buttons for execution, change and the saving of the stored commands.

The command "**Delete**" deletes the local storage file of the MDI commands (not the one saved on the CNC).

The command "**Change**" launches the program editor configured with the configuration panel for changing the stored command file.

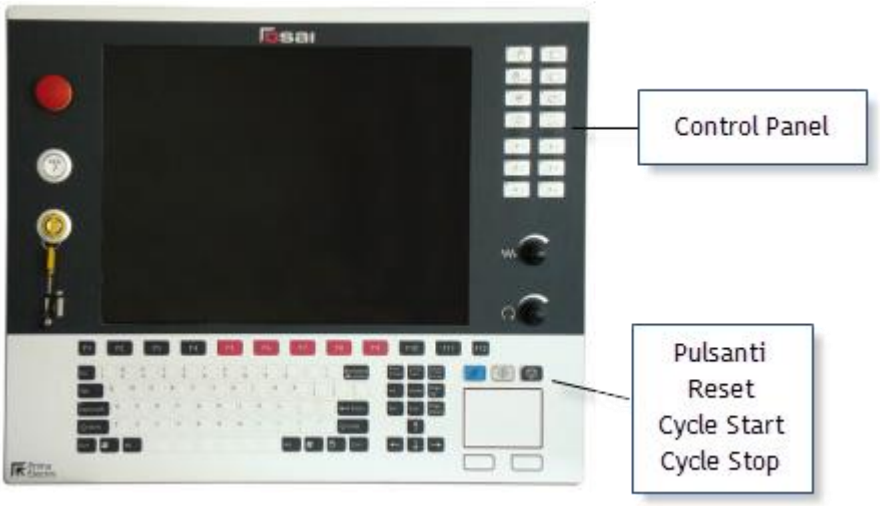
The command "**Save as…**" saves the commands buffer to the CNC and gives them a name.

The command "**Execute**" starts the commands buffer execution as a pseudo-program. The buffer is executed as a sequence of MDI commands sent from the ProcessController to the CNC. The pseudo-program. execution can be interrupted by RESET, MODE change, STATUS change (ie hold) or error during the execution of a block.

3.9 Operator Consoles

The Operator Console provides the operator with three control buttons for executing **Cycle Start, Cycle Stop** and **Reset** commands, plus a series of buttons and/or selectors for changing the speed of the axes and the mode.

3.9.1 Keys and functions of the OPENconsole COMPACT Operator Panel



As shown in the above figure, on the right side of the OPEN*console* COMPACT operator panel there are the operator console keys and the pushbuttons START, HOLD and RESET, i.e. the three control buttons described in the section "Control Buttons" at the beginning of this chapter. The buttons on the operator console are described below. For their functionality, refer to Chapters 5 and 6.

Enables continuous manual movements (JOG).

Enables incremental manual movements

Enables the axes homing cycle

Enables handwheel movements

Enables part-program execution in automatic mode

Enables block by block execution of the part program

Selects the MDI mode. The data entry input opens for keyboard entry of part-program blocks.

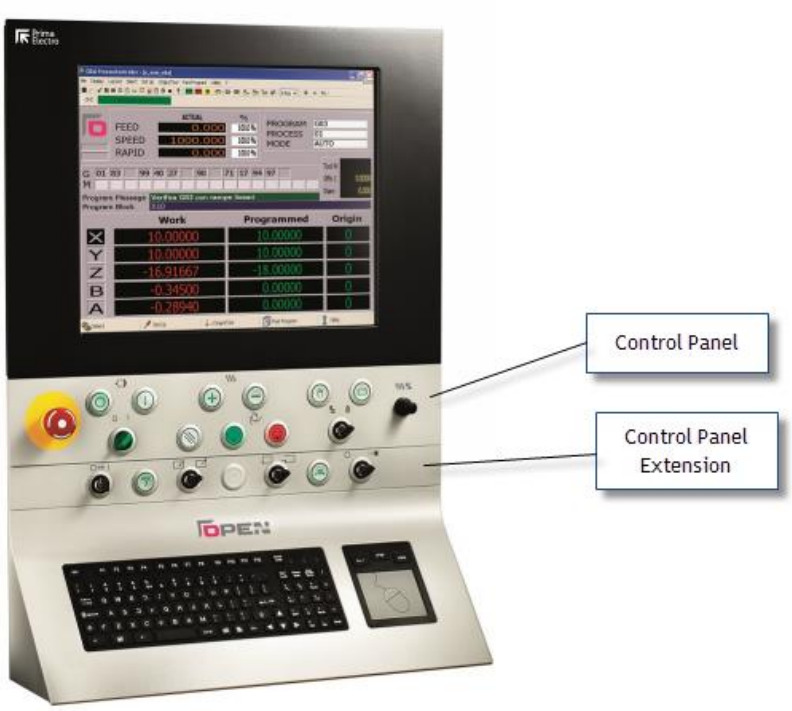
Rotary switch for the percentage change of the movement feed of the axes in automatic (Feed Rate Override) and manual (Manual Feed Override) modes.

Rotary switch for the percentage change of the spindle speed (Spindle Speed Override)

The buttons from P1 to P6 can be customized with functions set by the OEM. The Cycle Start, Cycle Stop and Reset buttons are described at the beginning of the chapter.

The functions of the buttons described here, are the standard ones managed by the system. They can be changed through the machine tool builder's machine logic. Refer to that documentations.

3.9.2 Keys and functions of the OPENconsole COMPACT Operator Panel



The OPEN*console* is a modular Operator Panel, optimized for machine tools mainly used in Automatic and manual continuous. The control panel can be extended with the module below that can have buttons customized for the application.
Below are described the keys of the standard operator console (for their functions refer to chapter 5 and 6).

Enable execution of programs in continuous automatic mode

Enables the continuous manual movments (JOG).

Moves the selected axis, in continuous manual mode, in the positive direction.

Moves the selected axis, in continuous manual mode, in the negative direction.

Drive ON.

Drive OFF.

Rotary switch for the percentage change of the feed movement of the axes in automatic (Feed Rate Override) and in manual (Feed Manual Override).

The Cycle Start, Cycle Stop and Reset buttons are described at the beginning of the chapter.

4. Establishing origins and homing the axes

4.1 General

On the machine tool you can define several reference points that allow the calculation of the axes homing positions. The following parameters are part of the machine configuration and are set by the MTB:

- the ABSOLUTE ZERO is determined by the position of the micro switch which is placed on the physical axis during each cycle reset
- compared to the previous position, using the NULL OFFSET parameter, the MTB may require that the zero position of each axis is moved as the quantity set: at the end of the homing cycle, on the micro switch, the axis executes an additional movement. At the end of this movement, it displays its position as 0.
- compared to the previous position, using the HOME POSITION parameter, the MTB may require that the zero position of each axis is moved as the quantity set: in this case, the axis does not move physically, but its position displayed is no more 0, as it coincides with the value set having th sign changed.

At the end of the reset cycle and the activation of the parameters previously described, it is defined what is seen by the end user as HOME POSITION i.e. the position in which all the axes have a displayed position equal to 0.

4.2 Axis homing procedure

Usually, after the machine boot, you can move the axes in manual mode (if allowed by the machine logic written by the MTB), but you cannot launch a program during the execution: this will be allowed only after following (at least once) an axes homing procedure.

If the axes have an absolute transducer, at the machine boot the axes already have a set position referred to their HOME POSITION, therefore you do not need to perform the homing procedure described below.

The standard procedure for axes homing follows the sequence below:

- select the HOME mode
- select the axis you want to home
- press the CYCLE START button
- wait for the end of the axis homing cycle
- repeat the procedure for all the axes

If the MTB has set the mode MANUAL RESET, the operator must hold the CYCLE START button until the end of the loop otherwise this cuts off prematurely: to complete it will be necessary to press the CYCLE START. If you have set the mode AUTO ZERO, after pressing it you can release the CYCLE START button: to stop the cycle must eventually be pressed the RESET button.

In some cases, the sequence to be used for the axes homing is determined by the machine status: for instance, let's suppose the machine has to restart after an emergency that left the tool in the workpiece. In this case, the operator of the machine will decide which sequence should be used or if it's better to manually move the axes to retract the tool from the workpiece.

Typically, the MTB includes a complete sequence of automatic homing for all the axes, that is launched in execution via specific commands: for this reason it is necessary to refer to the instructions of the MTB. The procedure previously described is still valid and can be used as an alternative to automatic one.

When the homing cycle of an axis finishes without errors, the message below appears:
22/6001 Axis homing complete
Distance micro-marker…

The value displayed for each axis in the previous message, indicates the distance between the release position of the micro switch and the position of the market transducer: the MTB sets the axes homing parameters so that the value is equal to about half of the mechanical pitch.
The message allows you to check this: if this value is too close to 0 or too close to the mechanical pitch, it could lead to a displacement error of the axis during the homing cycle equivalent to a mechanical pitch, with consequent errors in the manufacturing. Typically, the change of this parameter is due to mechanical causes such as the displacements of micro switch or a slip of the motor shaft.

4.3 Definition of the origins

Usually the MYB defines some reference positions different from the HOME POSITION, in various points of the machine (for instance, in the angles of the machining plane). In other cases, the final user needs to define additional positions referred to other machine points, according to his requirements. All these positions are defined as ORIGINS.

The OPEN*control* CNC allows the definition of up to 100 ORIGNIS for each process, identified by a number from 1 to 100. The number 0 identifies the HOME POSITION origin. Each origin includes a group of values (one for each machine axis) considered as the distance of the axis from his zero position: of course, this value can a positive or negative sign.

The user can use three different types of origin:

ABSOLUTE These are the origins referred to the zero machine (HOME POSITION) and are stored in the Origin Table. During machining, the program can recall at any time a certain origin (via the command UAO, indicating number indicating the identification origin number) and from that moment all the positions will refer to this new origin. For each process can be defined up to 100 origins. For additional details, check the Programming Manual.

TEMPORARY These are active origins, enabled by part program via the UTO command, referred to an absolute origin of which you have to temporary modify the value of each axis, of the quantity indicated (without changing the values stored in the origin table). These remain active until the calling for a new temporary or absolute origin or until the control reset. For additional details, check the Programming Manual.

INCREMENTAL These are origins referred to the active origin in that moment (absolute and/or temporary), defined and enabled directly from part-program using the UIO command. Unlike the temporary origins, where the following overwrites the previous one, the incremental origins programmed in sequence are summed. These remain active until the calling for a new temporary or absolute origin or until the control reset. For additional details, check the Programming Manual.

As aforementioned, the values to be inserted in the origin table for each axis, are considered as the distance of the axis from its HOME POSITION. If you want to move the axis origin towards its positive direction, the value to be inserted must have a positive sign, otherwise is negative. If the value is set =0, the axis origin coincides with its HOME POSITION.

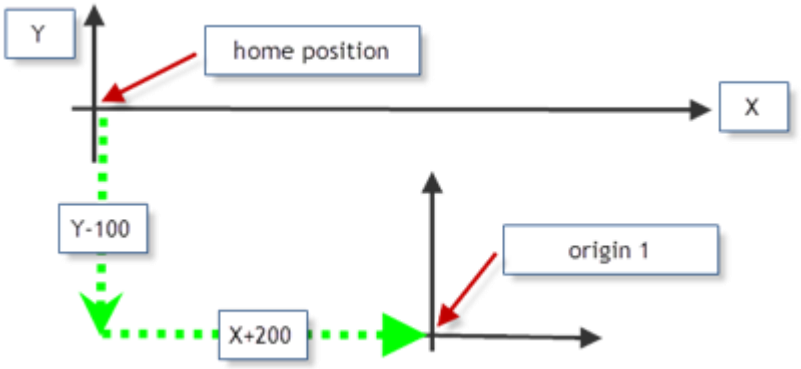
To set an absolute origin in the table, the user can use one of the procedures below:

1) manual setting of the absolute origin
2) automatic setting of the absolute origin

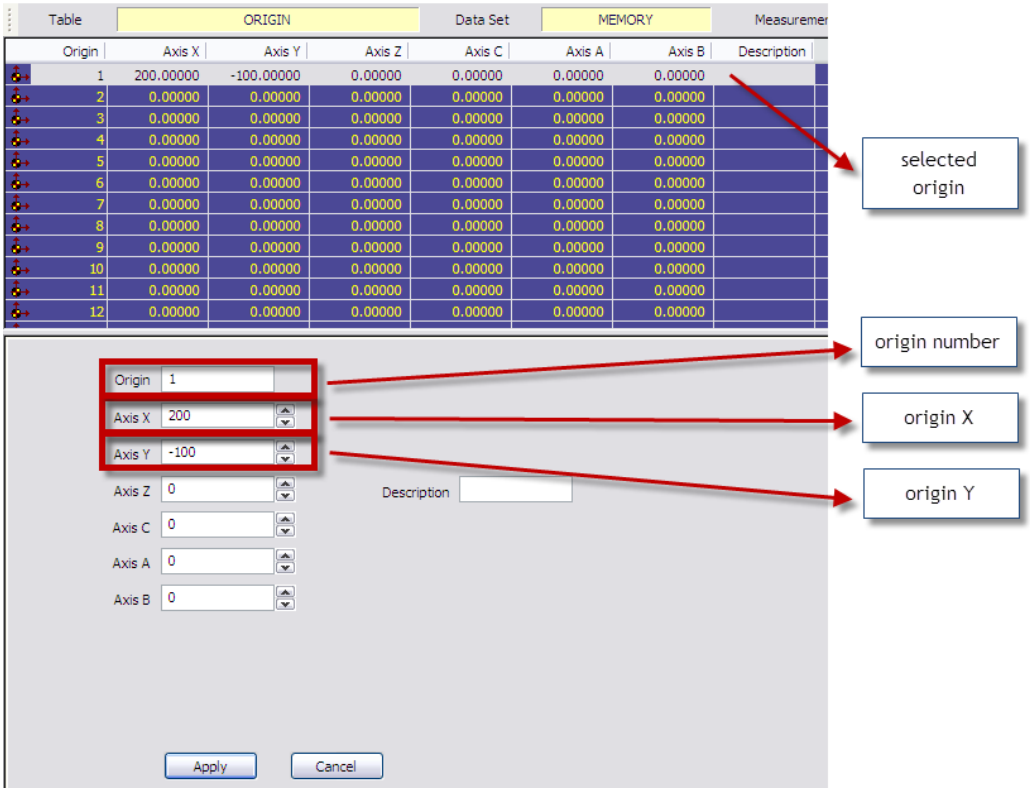
4.3.1 Manual setting of the absolute origin

In this case, it is assumed that the user already knows the value to be set for each axis and that these values should only be stored in the CNC working area (Origin Table).

For example, you want to set the origin 1 moved of +200 in the X direction and -100 in the Y direction, compared to the home position (see figure below).



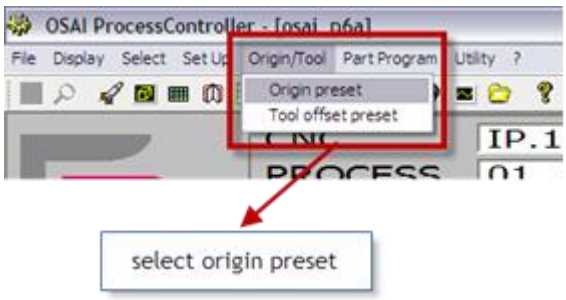
In order to get this, after opening the origin table and selecting the origin 1, you insert a +200 value in correspondence of the X axis and -100 in correspondence of Y axis. If you press APPLY, the values entered are stored in the origin 1.



In this example, for the axes (ZCAB) a null origin remains set, i.e. their zero position coincides with the HOME POSITION.

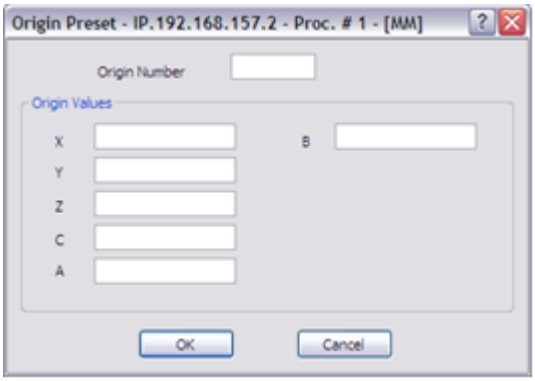
4.3.2 Automatic set of the absolute origin

The same change made for origin 1 can be automatically obtained, i.e. without opening the origin tables editor. Without activating any origin (so that the axis positions are referred to the HOME POSITION), physically set the X and Y axes in the positions desired (X200 and Y-100 , in our example). Select ORIGIN/TOOL in the tool bar and then ORIGIN PRESET.

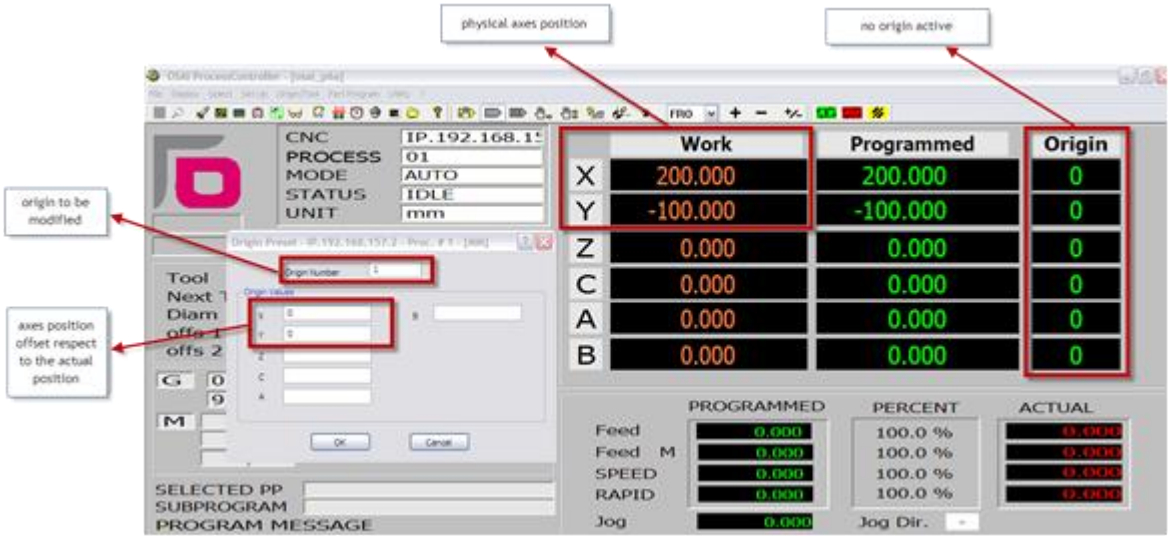


In the ORIGIN PRESET window displayed, you have to set:

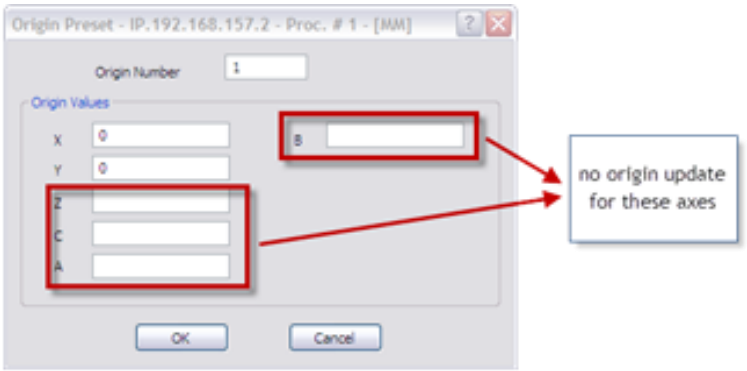
- ORIGIN NUMBER: the number of the origin you want to change (1, in our example)
- X..: indicated axis offset with respect to the present axis position



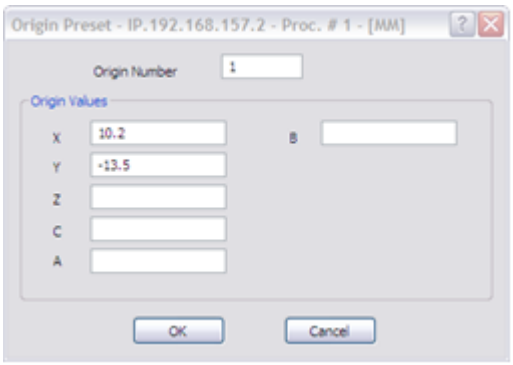
If you set “0” as the quote desired for the XY axes and then press “OK” and confirm, the system takes the present axes position (X+200 and Y-100, in our example) and writes it as the value of the origin 1 in the table. Therefore the origin 1 is physically updated with the value of the present position of the two axes: you can verify this opening the origin editor and checking the values of origin 1 for the X and Y axes.



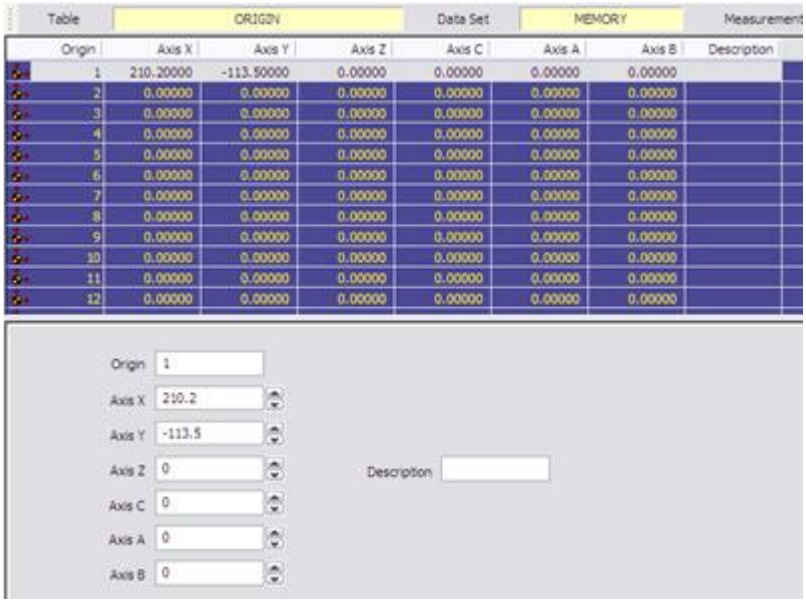
For those axes where there is no value set (axis Z, C, A and B in our example) the corresponding values set in the origin 1 <u>are not changed</u>.



If in the ORIGIN PRESET window you insert values different form zero, as the example in the figure below:



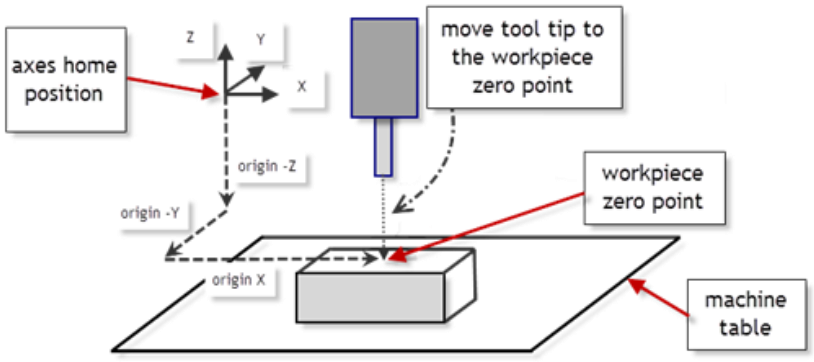
after confirming ("OK" button) the origin 1 is updated with the value (200+10.2) = 210.2 for the axis X and (-100-13.5) = -113.5 for the axis Y, as you can verify opening the origin table editor.



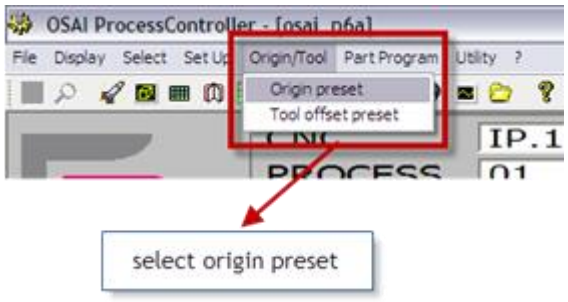
4.4 Axes and part origins

The part program has programmed positions referred to a reference position name PART ZERO or PART ORIGIN. For a machining program execution, the axes origin must coincide with the origin of the part.
A simple procedure is described below:

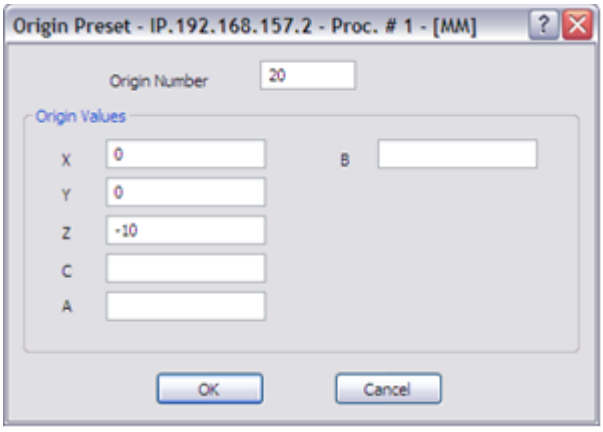
- place the raw part in the desired working position on the working plane of the machine
- enable the tool offset mounted on the spindle
- disable any origin so that the axes positions are referred to the HOME POSITION
- manually move the machine axes until the tool tip is on the point zero of the part (the positions displayed on the screen represent the XYZ axes positions with respect to HOME POSITION)



- on the tool bar, select ORIGIN/TOOL and then ORIGIN PRESET


OSAI ProcessController - [osai_p6a]
File Display Select Set Up Origin/Tool Part Program Utility ?
Origin preset
Tool offset preset
select origin preset

- set the required data in the ORIGIN PRESET window



In our example, the origin 20 is set with 0 value for X and Y axes, while the Z axis has a -10 offset so that the machining is made with a -10 depth with respect to the upper surface of the raw part (present position of the tool tip).

- after the confirmation, the axis positions are stored in the origin 20 of the origins table.

With this procedure, the origin 20 is correctly set and the part program can be sent to execution by putting it in the activation of the source using the command.

(UAO,20)

The previous procedure can be made also without having the tool mounted on the spindle. In this case, you will make the spindle nose coinciding with the zero position of the part. Obviously, no tool offset must be activated.

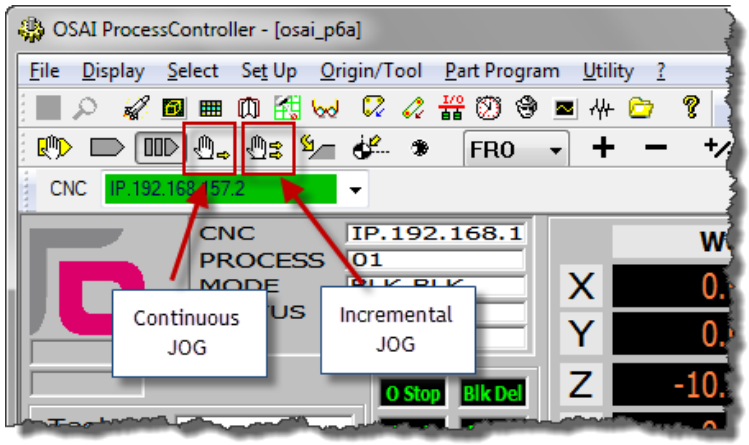
5. Axes jogging and Stop functions

5.1 JOG

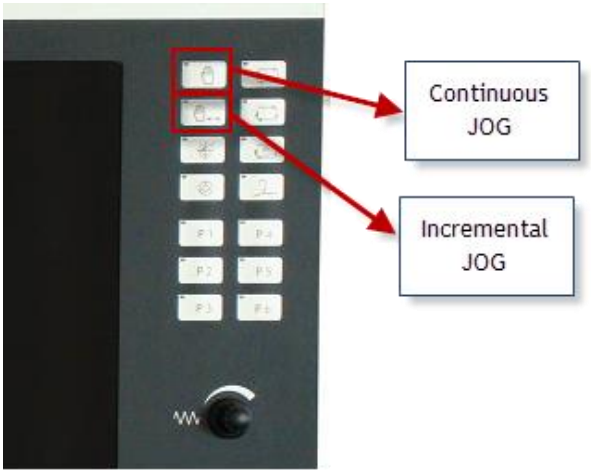
The manual movement (JOG) of an axis can be done in two ways:

- continuous jog
- incremental jog

To select a jog move (continuous or incremental), press one of the two buttons in the software Operator Panel (bar selection of the operative mode) of the ProcessController.



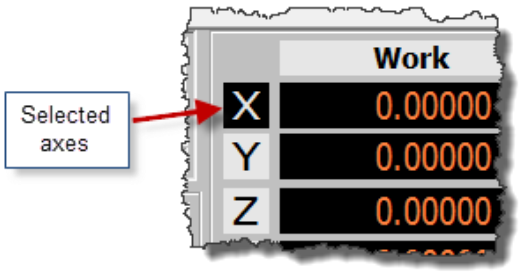
The same selection can be done on the OPEN*console* COMPACT Operator Panel using the selection buttons:



On the OPEN*console* Operator Panel, enable the manual movements using the button and select the continuous or incremental mode via the ProcessController.

To select manual mode the key-switch must be turned to the position indicated by the open lock.

The axis selected for the manual movement is the one shown in reverse on the screen:

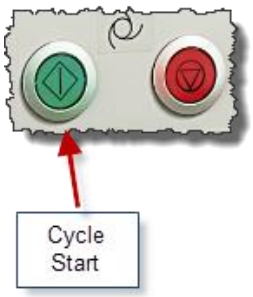


As long as the axes are not homed, the machine software overtravels are disabled. Before doing any manual movement, it is necessary to carefully consider what movement will be done. The machine tool builder handles the management of any hardware overtravel.

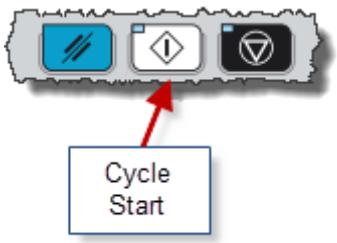
5.1.1 Continuous jog

This mode permits the axes to be jogged continuously in the selected direction at the selected rate. The procedure is as follows: (for the execution of each command see Ch. 3):

1. Select continuous Jog mode
2. Select the axis to move
3. Select the direction of the manual movement
4. Select the manual feedrate change percentage as shown in the section "Feedrate change" of this chapter.
5. Press the button "***Cycle Start***" and hold. As long as the "***Cycle Start***" button is pressed, the axis moves in the direction and the speed defined.



Cycle Start button in the OPEN*console* panel. The button could also be white.



Cycle Start button in the OPEN*console* COMPACT panel.

5.1.2 Incremental JOG

This mode allows the jogging of the axes for a pre-set increment in the selected direction at the selected rate. To move an axis incrementally, use the procedure below: (for the execution of each command see Ch. 3):

1. Select the incremental Jog mode
2. Select the axis to move.
3. Select the jog direction
4. Select the manual feedrate increment as shown in the section "Speed change" of this chapter .The jog increment is set with the data-entry enabled by the softkey command . "***Setup***"-"***Manual***".
5. Set the increment value. If this step is omitted, the system will apply the current increment, i.e. the latest one selected and displayed on the screen.
6. Press "Cycle Start".

There are two possible incremental jog modes which depend on the JOG INCR selection in the "Setup"-"Manual data-entry window:

- **Manual:** hold "***Cycle Start***" button until the move is completed. If "***Cycle Start***" is released before the end of the move, the axis decelerates to a stop. To continue the move press and hold "***Cycle Start***".

- **Automatic:** in this mode if "***Cycle Start***" is released the move will be completed without the axis stopping.

5.2 Manual Feedrate change

The default manual feedrate, defined by the configuration, can be changed with the following procedure (for the execution of each command see Ch. 3):

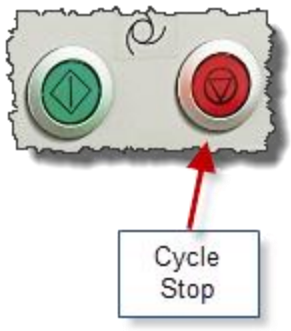
1. In the software operator panel, use the mouse to enable the button to select the override and select FMO (Feed Manual Override).

2. Using the buttons "+" and "-" to the right of the selector right, increase or decrease the current override value. Every time one of the two buttons on the screen are pressed, the percentage increases or decreases by 1.25%. The manual feedrate change has a range between 0% and 125%.

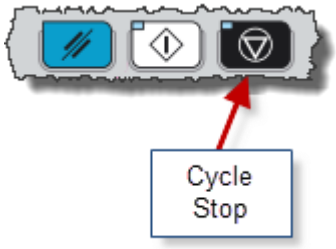
Example:
If the value configured for the maximum manual feed is 2000 MMPM (millimetres per minute) and a 50% percentage change, the actual manual feed will be 1000 MMPM.

5.3 Hold entry and JOG return on the profile

During part program execution, it is possible to stop machining by using the "***Cycle Stop***" button that puts the system in HOLD status. In this status, the CNC allows the axes to be jogged to remove the tool from the profile.



Cycle Stop button on the OPEN*console* panel. The button could be black.



Cycle Stop button on the COMPACT OPEN*console* panel.

In order to restart the part program execution, chose one of the following:

1. **Repositioning of the axes to the point of interruption of the profile**
   Operating mode:
   a) Select the axis to be returned to the profile.
   b) Select the desired feed percentage of the axis.
   c) Select Jog Return
   d) Press "***Cycle Start***".
   - if JOG RETURN is in "Manual" mode, keep "***Cycle Start***" pressed until the "AXIS ON PROFILE" message appears; this indicates that the axis return move has been completed.

   - if the JOG RETURN is in "AUTO" mode, just press "***Cycle Start***". The system will automatically move ALL the axes previously displaced by following the same trajectory in the reverse direction, so as to avoid obstructions. To abort automatic return to the profile press "***Cycle Stop***".

2. **Move the axes back to the final point of the interrupted block**
   Operating mode:
   a) In the Data Entry activated by the Softkey command "***Setup***"-"***Manual***", select the "Skip" option and confirm with "***OK***".
   b) Quit the HOLD status by pressing "Cycle Stop".
   c) Resume program execution by pressing "Cycle Start". At this point, the axes will perform a linear movement from their current position to the end point of the movement interrupted in HOLD. At the end of this positioning process, program execution will be resumed automatically.

This procedure cannot be performed in the following conditions:

- G61 active.
- Canned cycles and probe function on.
- TCP with mno/ijk programming or with rotary axes in "following" mode (interpolation with following mode).
- Program block in Hold without axis movements.

In such cases, the standard execution is forced and it is necessary to move the axes back to the profile interruption point according to procedure 1.

5.4 Spindle Speed Override

The value of the programmed spindle speed in execution can be changed with the following procedure (for the execution of each command see Ch. 3):

1. Click on the button to select the override in the software operator panel and select SSO (Spindle Speed Override).

2. With the buttons "+" and "-" to the right of the selector, increase or decrease the override value. Each time one of these softkeys is depressed, a 12.5% increment or decrement will be applied to the programmed value. Spindle speed override percentages range from 75% to 125%.

Example:
If the maximum programmed spindle speed is 1500 RPM (Revs Per Minute) and 80% speed override is applied, the actual spindle speed will be 1200 RPM.

5.5 Stop functions

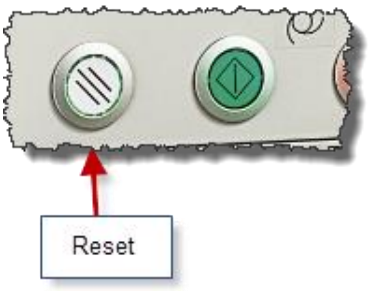
There are various ways of stopping program execution and machine operation.

5.5.1 Reset

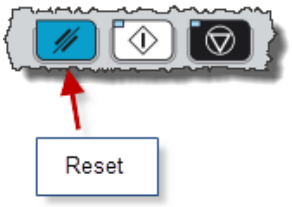
The reset function:

- Stops axes moves.
- Clears the program execution buffer.
- Restores the absolute origin (Home Position).
- Restores the configured G codes.
- Positions the part program to the start of the file.

To perform a reset function press "***Reset***" on the operator panel.



Reset button in the OPEN*console* panel. The button could be blue.



Reset button in the OPEN*console* COMPACT panel.

Press ***Cycle Start*** to resume execution**.** The execution restarts from the beginning of the part program.

Depending on how the machine logic has been programmed, it may give some typical stop commands (such as spindle stop, coolant off, etc.) when it receives the reset command. Check the MTB manual for details

5.5.2 Hold

The hold function:

- Stops the axes with controlled deceleration.
- Temporarily stops program execution.

To put the axes into hold press "***Cycle Stop***".

To resume execution press the ***Cycle Stop*** pushbutton again to exit from the hold condition and then press ***Cycle Start***. Execution will restart from the point at which the ***Cycle Stop*** command was given.
If manual moves have been performed after pressing ***Cycle Stop***, it is necessary to re-position the axes on the profile before resuming automatic execution. To do this, use the procedure described above.

5.6 Emergency Stop

The "***Emergency Stop***" pushbutton stops power supply to the system. The "***Emergency Stop***" pushbutton is not managed by the software system of the CNC and is controlled by the machine tool hardware. It is mounted on the Operator panel or in other parts of the machine tool.

5.7 Active reset

This command is available only in the WinPLUS logic program. It can be used after a cycle stop in order to cancel either the remaining portion of the current motion block or the subsequent part program blocks.

To allow automatic active reset, with WinPLUS the active reset function is available on an external pushbutton or on machine devices. For example, if a gap elimination device detects the contact between the tool and the part.

a) it resets the interrupted move.
b) searches the subsequent program block for execution. This block will be calculated from the current position.
c) executes assignment blocks, three-letter code instructions, branching instructions, etc. during the search.
d) after the subsequent move has been terminated, the ACTIVE RESET function is reset and the system does not re-launch the part program.

If there is a part program selected, the active reset command is allowed if the system is in IDLE status and the procedure is the same as those described above apart from point a).
If an active reset command is given when the system is neither on HOLD nor in IDLE status, the following error message will be displayed: "Command not congruent with the system status".

Active reset can be given in AUTO, MDI and BLK/BLK. If the control is in any other mode, the following message will appear: "Bad selected mode". If MDI (Manual Data Input) mode is active, active reset will only reset the interrupted move. All the instructions that have not been transferred to WinPLUS will be lost.

if active reset is allowed but the interrupted block is not a point to point move with linear or circular interpolation, the "active reset not permitted" message will be displayed. if the interrupted block has been stopped at the end of interpolation, the active reset command will be ignored.

The subsequent block must be a linear move (G1); if it is not, the "Active reset not permitted" error message will be displayed. If the subsequent block is a contouring move (G27/G28), the active reset function will be allowed but the new block will be executed as a point to point move.

All the M, T, S and logic functions that are programmed in the interrupted block or in subsequent blocks and have not been transferred to WinPLUS will be lost.

The new block calculated by the ACTIVE RESET command contains the coordinates of the axes programmed in the subsequent block. If the active reset command has been executed successfully, the system switches from HOLD to HOLD RUN (HRUN). The red LED turns off and the system awaits a "***Cycle Start***" command.

If an active reset function generates an error (for example, if the cycle start coordinate coincides with the subsequent motion block) the system remains in hold (red LED ON) but part program execution does not resume when you give a "Cycle Start" command. All the subsequent active reset functions will be allowed and the error will be removed.

Since during an active reset the system reads and executes part program blocks, some of the errors displayed may be numerical control errors. however, the "end of file" message is not an error but a signal. When this message occurs, the system status does not change to HRUN and "Cycle Start" permits execution of the block that has been restored after one or more active reset commands.

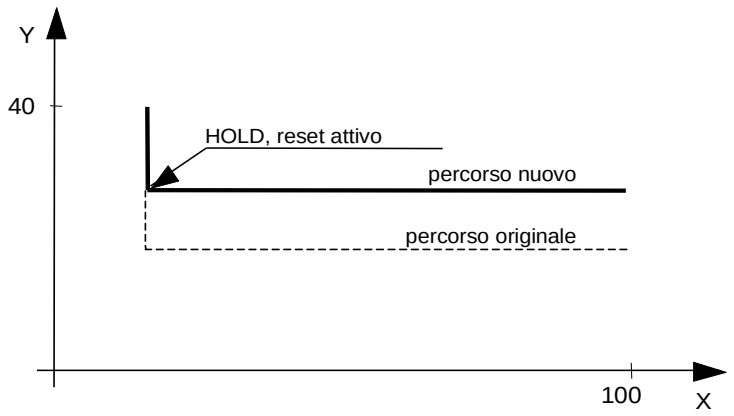
When the active reset function is not accepted by the system, further active request commands shall be taken to refer to the following blocks and, in the absence of errors, the active reset function for such blocks shall be accepted.

The block-end M functions programmed in the interrupted block will be "forgotten" after an active reset. Execution of the subsequent block will start when the "Cycle Start" command is given. If other active reset commands are given, the M codes programmed in these blocks will also be ignored.

G29 G01 X100 M05 Active reset at X=50 (M05 will be ignored)
G01 Y100 M06 Active reset (M06 will be ignored)
G01 M03 Y100 M05 Cycle start :
1. M03
2. Move to X50 Y100
3. M05

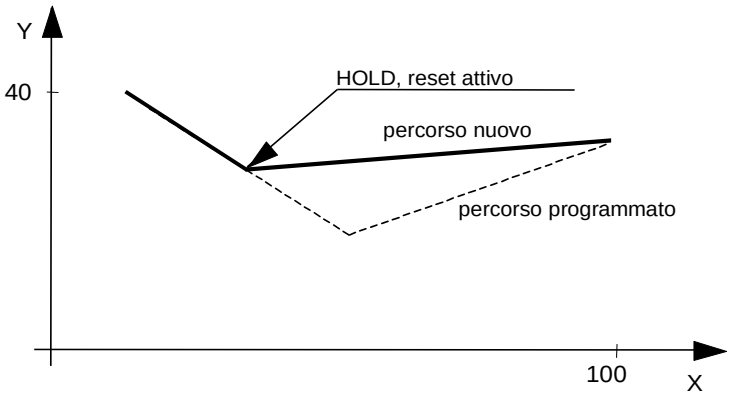
Any active reset function accepted by the system will enable the subsequent block. This can be observed on the display, where the enabled block will appear in reverse mode.

Example 1



X10 Y40
Y20 Active reset during execution of this block
X100 Y30 X100 at the end of this block

Example 2

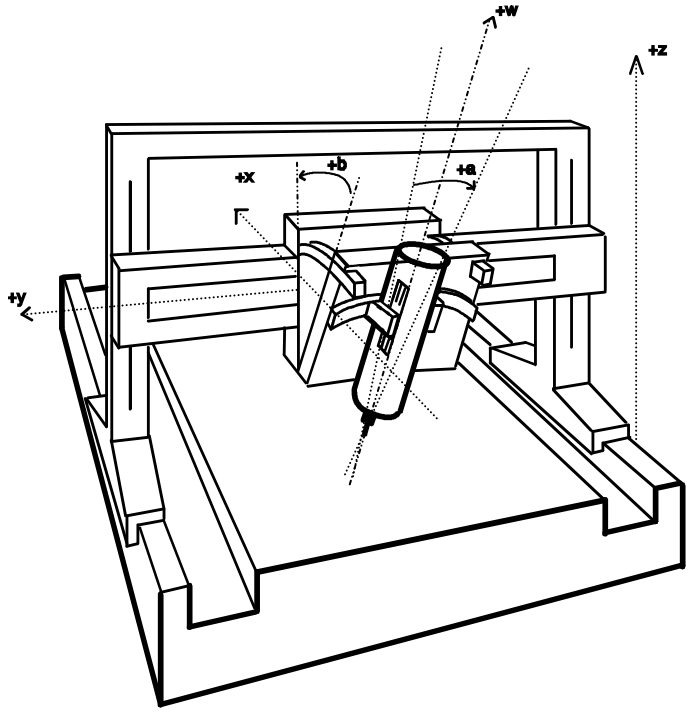


X10 Y40
X50 Y10 Active reset during execution of this block
X100 Y30 Y30 X100 at the end of this block.

5.8 Removing the tool after an Emergency

In some special applications (such as a 4- or 5-axis machines performing inclined machining in which the moves in the direction of the tool axes are the result of interpolations by other axes), execution may be interrupted by an emergency stop (voltage drop, general emergency, etc.) or by a programmed stop followed by manual moves.

An example of this is shown in the picture below.



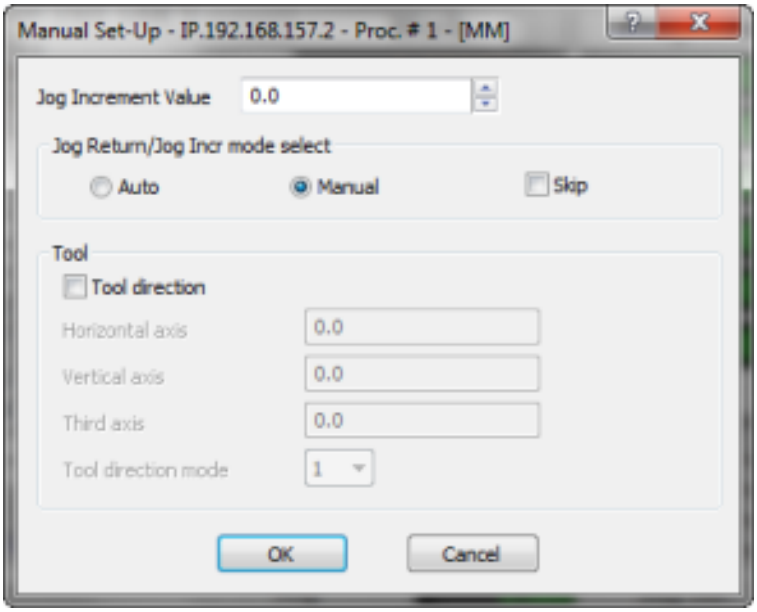
This machine has three normal axes (X, Y and Z) and two rotary axes that incline the spindle carrier head.

When an emergency stop occurs, it sometimes happens that the tool remains inside the part. To remove the tool from the part after the machine has been re-powered up, the system must offer you the possibility to move the spindle in the tool direction. This can be useful in case the tool remains stuck in the part during machining.
The system automatically defines the virtual axis whose name must be previously characterised in AMP. Movement of this virtual axis causes the linear axes to move, so that the tool moves in the linear axes direction, i.e. in the direction indicated by the position of the rotary axes.

The generation of the virtual axis along which the axes movement is required is set via the "Tool direction" field in the data-entry "***Manual Setup***" enabled by the softkey command "***Setup***" – "***Manual***".

The procedure is:

1. Open the data-entry “Manual Setup”:



2. Tick the field “***Tool Direction***”
3. If necessary, change the positions related to the (max) three rotary axes.
4. Check and if necessary correct the positions of the rotary axes and the selection of the Tool Direction Mode
5. Confirm with “***OK***”. At this point, the system generates the virtual axis for the movement of the axes along the tool direction or the axes of its plane (see below).

The three fields are editable, allowing e.g. to change the direction of tool output or input the correct positions of the axes that may have been moved between the time of the anomaly and when the "Tool direction " mode is activated.

Horizontal axis
Value in degrees of the position of the horizontal rotary axis (described as “Work rotary axis” in AMP)

Vertical axis
Value in degrees of the position of the vertical rotary axis (described as “Tool rotary axis”.in AMP)

Third axis
Value in degrees of the third rotary axis (described as “Middle rotary axis”.in AMP).

The values of the three fields can be changed only before the execution of the cycles reference axes.

Tool Direction Mode
Allows enabling of the axes movement along the tool direction and its reference plane. In AMP three virtual axes are configured (see Kinematics), one along the tool direction and two related to the plane fixed to the tool direction.

The value 1 only enables the tool direction axis.
The value 2 enables both the axes.
The value 3 enables all the axes, but the reference plane does not comply with the tool direction, but complies with the machine plane.

NOTE:
If only one of the rotary axes is referred, the system displays the position corresponding to the one not referred: the previous considerations remain valid. See also the Tool Centre Point, mode 5, in the Programming Manual.

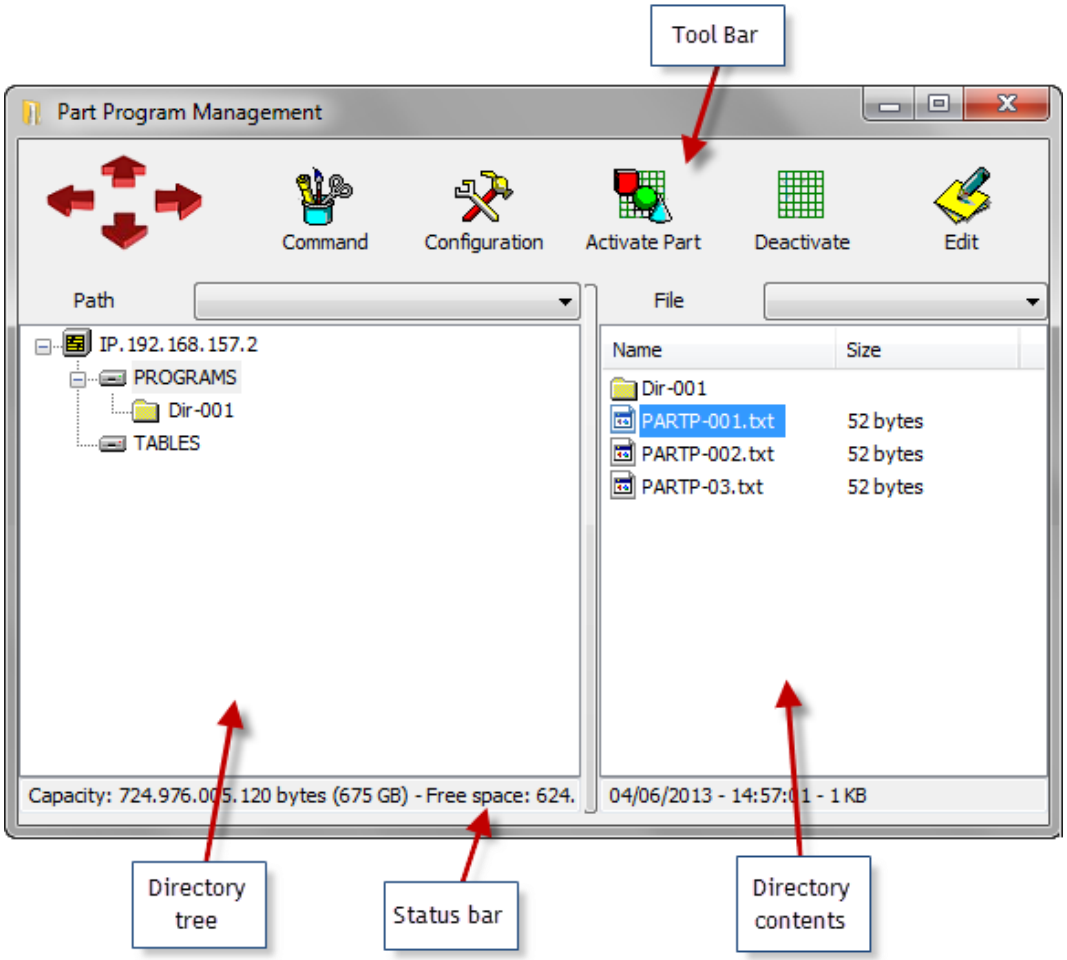
The functions described in this chapter are the system standard. They can be changed by the machine tool builder's machine logic. Refer to the MTB documentation.

6. Part Program Management

The OPEN*control* CNC has an ambient dedicated to the part program file management, that can be distributed in several directories. The program management ambient is therefore the file manager that activates the program and also allows the management of the files and system directories.

6.1 Main Video Screen

Pressing the "**Part Program**" softkey of the *ProcessController* and then selecting "**Part Program Management**", the ambient screen video is displayed:



The video screen is divided in 4 functional areas:

- Directory Tree
- Directory Contents
- Tool Bar
- Status bar

"Directory Tree" Area

As a reversed tree, in this area are displayed the devices and their content. Starting from the root, each branch represents a device (CNC/GMC/PC). The sub-branches are the directories contained in the device. This area displays the files contained in the directories. The content of the selected directory (file or other directories)is displayed in the "Directory contents" area.

In this chapter, the area is also indicated as "left panel".

"Directory Contents" area
This is the area displaying the content of the directory selected in the left panel. It lists the directories and the files. Selecting a directory in the left panel, the content of this read is automatically updated.
In this chapter, the area is also indicated as "right panel".

Toolbar
It contains the command to be applied on the directories and on the files. The commands available in the toolbar are also available selecting a directory or a file with touch-pad right-click.

Status Bar
It contains the data referred to the selected element.
If a logic drive is selected in the Directory Tree area, the status bar below displays the capacity and the free space. If in the **Directory Contents** area is selected an element, the status bar below displays the time and date of creation for the directories; instead displays the date, time of creation and size for files.

6.2 Selection of an item

Usually, the several commands available in the *Program Management*, refer to the "**selected element**". The selected element is the *device*, the *file* or the *directory* displayed in reverse in the "*Directory Tree*" or "*Directory Contents*" areas. To select an item, you can use touch-pad and selection keys, those from keyboard or those graphic in the Toolbar of the *Program Management*.
If using the touch-pad, once the cursor is on the element, left-click.

6.3 Multiple selection

In the right panel it is possible to select several items. This allows the simultaneous application of a command to all the selected elements (for instance, the delete and drag&drop copy commands). A selected item is displayed in reverse.

The multiple selection, always done in the right panel, can be done is different ways:

Method 1, for close items, touch-pad selection:
- Select the first element using the touch-pad
- Hold the "Shift" button of the keyboard and select with the touch-pad the last element
- All the items included between the first and the last are selected

Method 2, for close items, selection with the arrow keys "***Up***" and "***Down***" on the keyboard:
- Select the first element using the touch-pad
- Press the "Shift" pushbutton and press several times the key "Up" or "Down". each time you press either button, a file form selection is added or removed (according to the selection of the movement with respect to the first element)

Method 3, for discontinuous elements:
- Select the first element using the touch-pad
- Press the "Ctrl" key and select with the touch-pad another element, not necessarily close to the previous one. If you select an element already selected, the element is removed from the selection.

All items selected individually, are added to the selection.

6.4 Creation of a new Part Program

To create a new part program:

- select in the left panel the directory in which you want to create a new file.
- right-click on the touch-pad in an empty area of the right panel. Select "***Create File***" from the drop-down menu.
- In the directory is created the "***NewFil##***" file. Where ## are two digits used to differentiate one file from the other in case more than one file is created without renaming them. The first file will be *NewFil01*, the second *NewFil02* etc.
- The field containing the name of the file just created is opened for editing. Write the new name and press "***Return***" on the keyboard, or select another element form the screen with the touch-pad and confirm the new name; to exit the editing and restore the original name, press "**ESC**" on the keyboard.

To write the program in the file, see "Edit" command

6.5 Creation of a new Directory

The procedure is very similar to the one used for the creation of a new file.
To create a new part program:

- select in the left panel the directory in which you want to create a new file.
- right-click on the touch-pad in an empty area of the right panel. Select "***Create Dir***" from the drop-down menu..
- In the directory is created the "***NewDir##***" file. Where ## are two digits used to differentiate one file from the other in case more than one file is created without renaming them. The first file will be *NewDir01*, the second *NewDir02* etc.
- The field containing the name of the file just created is opened for editing. Write the new name and press "***Return***" on the keyboard, or select another element form the screen with the touch-pad and confirm the new name; to exit the editing and restore the original name, press "**ESC**" on the keyboard.

To write the program in the file, see "***Edit***" command

6.6 File copy and directory

To copy a file or a directory:

- Search, via the left panel, the directory containing the element to copy and select it.
- right-click on the touch-pad in the right panel once the cursor is on the element name to be copied.
- Select "***Copy***"
- Using the left panel, select the destination directory.
- right-click on the touch-pad in an empty area of the right panel. Select "***Paste***" in the drop-down menu.

If the destination directory coincides with the one of the element to be copied, the element is copied and automatically renamed, adding in the end of the name a progressive number.
To copy on e or more files or directories, it is also possible to "drag&drop", i.e. using the touch-pad, drag the file name from the directory location to the destination one.
The procedure is:

- Open the destination directory in the left panel.
- In the left panel, search the directory containing the file and copy it.
- Left-click on the touch pad after placing the cursor on the file to be transferred, hold the pushbutton (the multiple selection can be also used to move several elements at the same time – see paragraph "Multiple selection")
- Using the touch-pad, drag the icon up to the destination directory in the left panel
- Release the touch-pad button.

This can be simplified by using the configuration screen in four quadrants. See the explanation of the "Quad View" command.

6.7 Delete a directory or a file

To delete a file or a directory:

- Using the left panel, search the directory containing the file or the directory to delete.
- right-click on the touch-pad, in the left or right panel, after placing the cursor on the element to delete.
- Select "***Delete***"
- If the element to be deleted is a directory, an additional confirmation to delete the content is required.

To delete a directory and several files at the same time:

- Using the left panel, search the directory containing the file or the directory to delete.
- Multiple selection of file in the panel (see "Multiple selection" paragraph).
- Press "***Delete***" on the keyboard.
- Before deleting the selected elements, the systems requires a confirmation.

The file and the directory are permanently erased. When deleting a directory, all its content is erased (files and sub-directories).

6.8 Change the name of a file or a directory

To change the name of a file or a directory:

- Using the left panel, search the directory containing the file or the directory to rename.
- Right-click on the touch-pad in the right panel on the item to rename and select "***Rename***". Alternatively, select the item with the left button of the touch-pad, wait 2 seconds and select it again.
- The field containing the name of the item passes in "***Editing***" mode. Write the new name and press the "***Return***" pushbutton on the keyboard or select another video item with the touch-pad to confirm the new name; exit the editing and restore the original name, press "**ESC**" on the keyboard.

6.9 Writing and editing a Part program

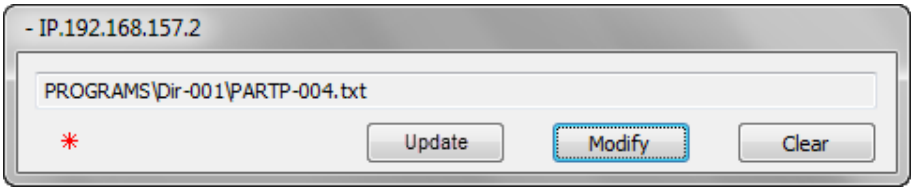
The part-programs are text files in ASCII format and can be edited with any Editor that does not change their format. The file can be defined or edited on the CNC or, offline, on a PC and then copied in the control. The Online Editor used by the CMC depends on the configuration set by the OEM using the WinNBI configuration application "NBIconfig" password-protected.

In the configuration released by Prima Electro the pre-set editor is Microsoft WordPad.
For the Editor use, refer to the documentation provided by the application provider.

The edit of a par program is always done on a copy of the program. The file opened for editing is previously duplicated (the file is moved from the CNC to the PC in case the system hardware configuration has a PC dedicated to the HMI or is copied in a temporary directory in case the configuration has only one CNC module).
This has two advantages:

- it is possible to work on the PC HMI system in the same way as on the system having only the CNC
- allows the file editing even if it is selected for the execution (see further on)

The Program Management includes a panel for managing the temporary file in editing (see picture below).



The "***Update***" button applies the changes made. It is enabled only when the date/time of the file in editing are changed compared to the original, i.e. after saving the file changes done with the editor. Enabling the button, the original file is overwritten with the temporary one and the panel is closed. This command cannot be completed if the file is busy with other programs. Same thing for the program enabled for the execution. Before updating the program in execution, close it.

The button "***Modify***" re-opens in editing, Pressing this button the editor re-opens to edit the temporary file. The button "***Clear***" resets the changes made to the temporary file. Pressing this button, the temporary file is deleted and the panel closed.

The program "*Program Management*" can be closed also if the panel to manage the program editing is opened. When re-opening the "*Program Management*", the panel is displayed to complete the editing operations.
In the panel, the button to be used by the user is blinking. Until the file has not been edited, the button "**Modify**" blinks, following the first edit, the button "**Update**" blinks. Therefore, when the "*Update*" button is blinking, it indicates the file has been edited form the original one.

Follow this procedure to edit a part-program:

- If the file does not exist already, create it following the procedure "Creation of a new Part Program"
- In the right-panel, select the file to edit.
- Press "***Edit***" in the Toolbar.
- The part-program opens in editing using the configured editor.
- Edit the file, save it and close the editor.
- In the panel described above, overlapped to the Program Management, select "***Update***".
- The file is edited and updated on the CNC.

To execute several editing sessions on the same file, i.e. to re-edit a file once closed and before the updated execution, use the "***Modify***" button in the panel to manage the temporary file instead of "***Edit***".

6.10 Enabling a Part-Program for the execution

To execute a Part Program, it must be selected and enabled.
The CNC reads the part-program from the file selected, that remains busy in reading until the command "***Deactivate***" is released.

To enable a part-program:

- in the left panel, select the directory containing the file.
- in the right panel, select the file name
- press "***Activate***" in the Toolbar.

The name of the program activated and its firs lines, are displayed in the video screen of the ProcessController (if available in the selected video).

This procedure enables the program, but does not execute it.
The procedure to execute the program is described in the Ch.7 "part Program Execution".

6.11 Disabling the Part-Program selected

To disable the selected program for the execution, press "***Deactivate***" in the Toolbar. It is not necessary to select the file with the touch-pad to execute this operation. The CNC allows the deactivation of the program only in IDLE status.

6.12 Program Management Configuration

The Program Management works on the configured devices. The Program Management displays in the left panel all the devices configured and allows the execution of the operations described in the previous paragraphs (also between devices). For example, it is possible to execute the copy drag&drop of a file from a CNC to another of from a PC to a CNC and vice versa. By "device" is meant an OSAI CNC or GMC, or even a PC or a Teach Pendant.

The "Add device" command is available in a drop-down menu that is displayed by clicking the "Configuration" button in the Toolbar. This command only requires the network name of the device that you want to connect. When a device has been connected, the Program Management is able to view all the relative drives configured.

NOTE:
In the panel opened by "Add Device" command the network name or the TCP/IP address of the device must be specified adding the prefix "IP." For instance, if you want to connect an OPENcontrol CNC whose network name is "OPEN-XS_001" and whose network address is "192.168.1.26" in the device connection panel you may write either "IP.OPEN-XS_001" or "IP. 192.168.1.26".
To select a local device, i.e., the device where the Program Management is active is active you may use the prefix "IP."
Please note that a device can be a PC (local or remote) making possible to use it a Part Program repository from where the programs are copied to the CNC executing them.
When the PC is the one running "Program Management" simply add to the configuration the "IP." device to access the configured PC logical drives (see below).

By default the Program Management connect the device selected by ProcessController. The user is not required to configure this device.
The devices connected by the user with the "Add Device" command remain in the Program Management configuration on a permanent basis and are connected again each time the application is launched.

To remove a device from the configuration select the device name in the Directory Tree panel and press the "Configuration". In the drop down menu select the "Remove Device" command.
The visualization of device will be removed from the Directory Tree but the contents of the device will be unchanged.

6.13 Logical drive configuration

All drives shown for each device in the Directory Tree are directories that have been configured on the target device. The drives are simply a name assigned to a directory of the device. This is way they are called "logical drives".

Enabling the configuration gives access to commands that make it possible to add or remove logic drives on all the devices configured. It is possible to create logical drives on each device connected by "*Program Management*", also the remote one with the exception of the PC device. In the last case, for privacy reasons, to be able to create a logical drive, Program Management must run on the PC itself (see the warning below).

The "***Enable Configuration***" command is available in a drop-down menu that opens when you click "***Configuration***" button in the Toolbar.

To access the configuration you must enter the OEM password.
The predefined password of WinNBI is "OEM".
The possibility to change the configuration remains active, without the need to re-enter the password, until the Program Management is closed.

6.13.1 Adding a logical drive

The command "***Add logical drive***" is available in the drop-down menu that is opened by clicking the "***Configuration***" button in the Toolbar, when a device is select in the *Directory Tree*.
In the panel that opens activating the command, you must enter the logic name of the drive (which users will see in the Directory Tree) and the absolute route of the directory to be associated with the drive. The route must be valid for the device referred to. For example, "\SSD\USER\PROGRAMS" could be a valid route for OPENcontrol while "C:\PartPrograms" could be the valid route for a PC.
The specified directory must exist on the selected device.

The configuration resides in the (local or remote) selected device that is being configured, and is read by the Program Management when activated, and therefore the logical drives added to the configuration of a device are visible in the network by all the other admissible OSAI devices.

For data security purposes, it is NOT possible to configure a drive on a remote PC. The drives that you want to configure on a PC must be configured by launching the Program Management (or the "***FileBrowser***" WinNBI application) residing in the PC itself. In other words, only a user authorized to work on a PC may decide which directories, if any, will be visible to the other network devices.

6.13.2 Removing a logical drive

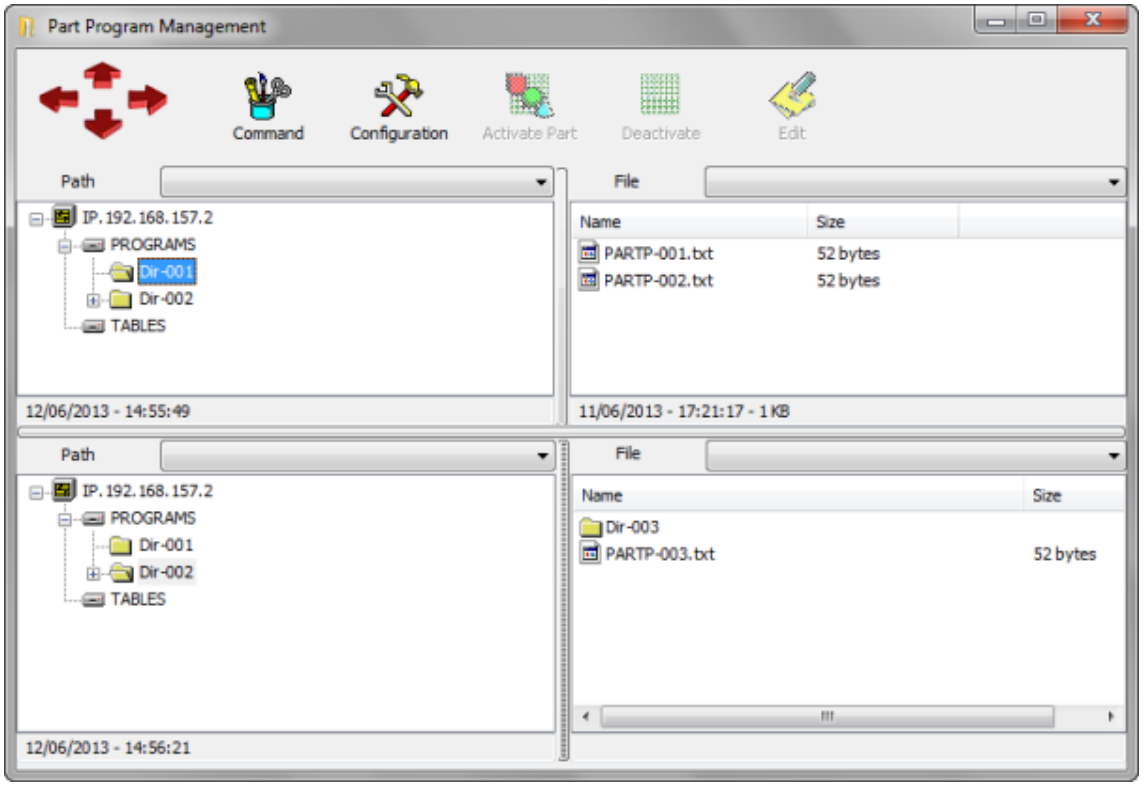
This "***Remove logical drive***" command becomes available only when the configuration has been enabled with the "*Enable configuration*" command.
The command is in the drop-down menu that is opened by clicking the Configuration button in the Toolbar when a drive is selected. Upon being confirmed, this drive is removed from the *Directory View* and from the configuration. All drive contents on the target device will remain unchanged.

6.13.3 Showing logical drive path

The "***Show logical drive path***" command displays the physical path configured for a logical drive.
This command becomes available only when the configuration has been enabled and when a drive is selected in the *Directory View* panel.

6.14 Four Windows Display (Quad View)

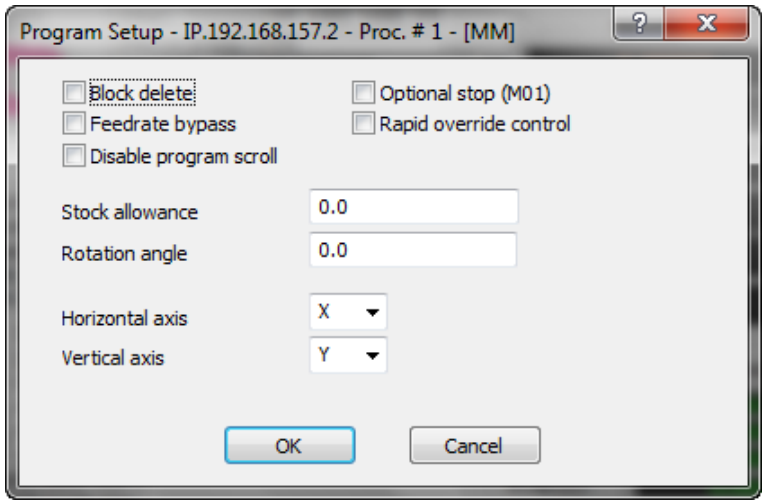
Using the "Quad View" command available in the list that appears when clicking "***Command***" in the Toolbar, , you switch the display mode to the Quad View:



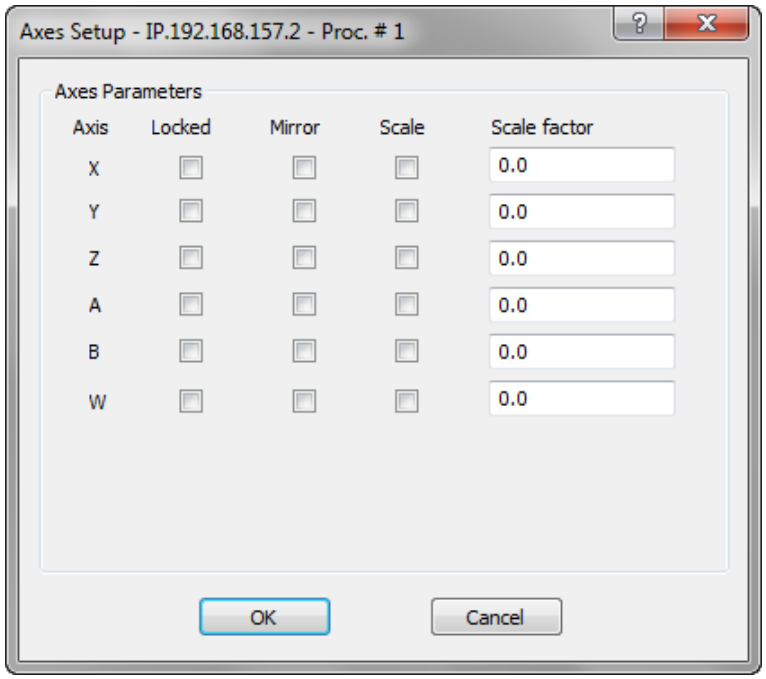
The two additional windows that appear have the same content of the previous windows.
For instance, the windows simplify the files drag&drop operations from a directory to another. For example, to execute this operation select, in the upper panel, the origin directory and in the lower panel the destination directory.

7. Part Program execution

Prior to executing a part program, it is possible to see a series of parameters that affect the execution mode and enable/disable controls, visualisations and special operations.
These parameters are available in two data-entry windows, the "***Program Setup***" data-entry, enabled by sending the softkey command "***Setup***"-"***Program***"



and the "***Axes Setup***" data-entry, enabled by sending the softkey command "***Setup***"-"***Axes***".



For the data-entry description, refer to Ch. 3 " User Interface".

7.1 Part program execution

There are two part program execution modes:

- Auto
- Semi auto (block by block)

In order to execute a part program it is necessary to:

1. Select and activate the part program
2. Select the execution mode
3. Launch execution.

7.1.1 Selecting and activating a part program

These operations must be carried out before selecting the execution mode. The procedure is as follows:

- Press the Part Program softkey and select the command ***"Activate Part Program"*** or **"Part Program Management"**.
- Select a part program for execution as described in Ch. 6.
- The name of the program selected and the first lines to follow are displayed on the screen.

7.1.2 Automatic execution

This mode permits to execute all the blocks in a program without interruptions. To execute the program in auto mode, use this procedure (for details on the operation see Ch. 3):

- Select and activate the part program
- Select the AUTO mode. The selection is displayed on the screen.
- Press the ***Cycle Start*** pushbutton.

The selected part program will be executed from the first to the last block.

7.1.3 Block-by-block execution

This mode permits to execute one part program block (program line) at time. Usually this mode is used to check if a new program is correct as, at the end of each program block, the execution is interrupted.

In order to execute a part program in the block-by-block mode, proceed as follow:

- Select and activate the part program.
- Select the "BLK/BLK" mode. The selection is displayed on the screen.
- Press ***"Cycle Start"*** each time you want to execute a block. Blocks will be executed in the same sequence in which they are programmed. After executing a block the system stops until the operator gives another *Cycle Start* command.

To alter the sequence of block execution or bypass one or several blocks, use the arrow keys to position the cursor to the desired block.
To close block-by-block execution, select another operating mode.

If BLK/BLK is pressed during the part program automatic execution, the system executes all the blocks already analysed by the CNC and then stops. To resume part program execution in block-by-block mode, press ***"Cycle Start"***.

7.1.4 Multiblock retrace

This function permits to retrace a series of programmed blocks by moving the axes backwards along the executed profile. Depending on the parameter configured in AMP, the number of retraceable blocks varies from 1 through 64.

The procedure is as follows:
Press the "***Cycle Stop"*** pushbutton to stop the axes

1. Using the ***"Setup"*** – ***"Multi Block Retrace"*** softkey command, open the "***Block Retrace***" data-entry window.

2. Enable the item "***Enable Block Retrace***" and confirm.

3. Press the "***Cycle Start"*** pushbutton. The axes will move backwards in the selected mode
   - In BLK/BLK mode the profile will be retraced one block at a time each time the ***"Cycle Start"*** pushbutton is pressed.
   - In AUTO mode the system will retrace the last n blocks in the profile. n is the number of blocks configured in AMP.

4. When the retrace move is completed the message **"End of multi block retrace"** appears.

In order to stop a backwards move, press "***Cycle Stop***" pushbutton. Now you can backwards move can be restored or the movement direction can be inverted. In the first case, press "***Cycle Start***" pushbutton. In the second case:

- Using the softkey command "***Setup***" – "***Multi Block Retrace***", open the "***Block Retrace***" data-entry.
- Enable "***Emit Auxiliary Func.***", if during the Block Retrace execution the auxiliary functions have to be released, stop it. If not, press "***OK***" and confirm.
- Press the "***Cycle Start***" button. The axes will move in the mode selected at the previous step.

NOTE:

- During part program execution in AUTO mode the "***Block Retrace***" function can be performed in BLK/BLK mode. Part program execution will be resumed automatically when the profile has been retraced and the axes have returned to their stop point.

- When the "***Block Retrace***" function is active the axes move in point-to-point mode (G29) in the backwords retrace, while in the forwards trace they are executed according to the programmed modality (G27,G28,G29).

- In "***Block Retrace***" mode, the control unit performs the motion blocks only and, on request, the emission of auxiliary functions; assignment blocks are ignored.

- The emission of auxiliary functions during the "***Block Retrace***" forward move can also be enabled by assigning the **MBA** process variable.
  - MBA = 1, emission is enabled
  - MBA = 0, emission is disabled

- When the "***Block Retrace***" function is active the feedrate can be altered with the Feedrate Override function (see later in this chapter).

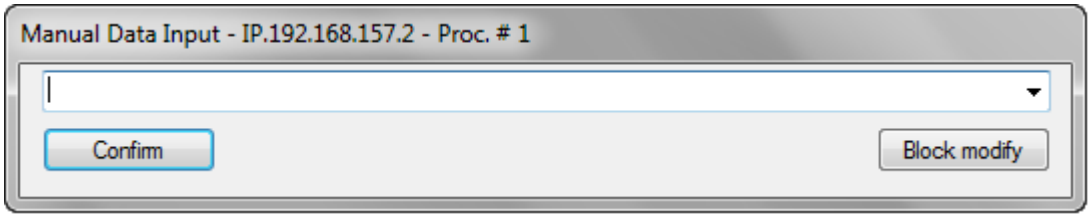
7.2 Executing blocks from the keyboard

The system permits to execute blocks entered from keyboard (MDI) both when a part program is active and when no part program is selected.

7.2.1 MDI execution without an active part program

The procedure is as follow:

1. Select the MDI mode. On the screen appears an input window to write the block to execute:



2. Write the block to be executed in the data entry window
3. Press "***Confirm***" to confirm the input block. As the block is confirmed, the MDI line displayed in the data entry window will turn green. The list of the last commands previously confirmed can be displayed and reused pressing the right arrow of the input field.
4. Press "***Cycle Start***" to execute the inserted block.

For th introduction of another block, repeat steps 3 and 4. To reset the MDI mode, select another operative mode.

7.2.2 MDI execution with an active part program

While the system is executing a part program it is possible to execute blocks in MDI mode. The input block does not affect the active part program and is not retained in the system memory.

To execute a block in MDI mode use this procedure:

1. Select the MDI operative mode. This operation stops the part program execution after executing all the pre-analysed blocks, if the program has not been stopped with the **BLK/BLK** softkey.
2. Write the block to be executed in the data entry window and press "***Confirm***" to confirm it.
3. Press "***Cycle Start***" to execute the confirmed block.

To input other blocks for execution repeat steps 2 and 3.
To restore part program execution select the "***Automatic***" operative mode or ***"BLK/BLK"*** and press the "***Cycle Start***" pushbutton. The program will restart from the interruption.

If **MDI** is pressed during automatic execution, the system executes all the pre-analysed blocks and then stops.

Execution of paramacros in MDI mode
This function makes it possible to run in MDI mode the paramacros called implicitly by the S,T,M, auxiliary functions, which must have been configured beforehand as paramacros.
For information on how to program these functions as paramacros, see the Programming Manual.

To run a paramacro in MDI mode, proceed as follows:

1. Activate the auxiliary function in MDI mode, as described in points 1 and 2 of the previous paragraph. This displays the name of the corresponding paramacro in the list of active programs.
2. Deactivate the MDI mode by pressing the MDI softkey again.
3. Press the [CYCLE START] button to execute the paramacro.

At the end of the execution, the paramacro is closed and the system returns to MDI mode.

NOTE:
If you try to activate the same paramacro several times, the error message "programme already selected" (error 149) appears.

7.2.3 Executing part of a part program

The partial execution of a part program is the execution of the instructions between two defined blocks. When part program blocks are numbered the system can execute the part of program delimited by two block numbers.

The procedure is as follows:

1. Select and activate the part program.
2. Open the data-entry using the "***Part Program***" – "***Exec From To"*** softkey command
3. Insert the starting block number, the label or the number of the file line where starting the execution
4. Select and fill in the fields (block number, label, number of the file line or $BRK instruction) of the conditions determining program stop execution.
5. Press "***OK***" to confirm data
6. Press "***Cycle Start"*** to start executing the part of the program delimited by the specified blocks.

7.3 Modifying blocks in BLK/BLK mode

Part program blocks can be altered and executed from the keyboard. These alterations do not affect the original programmed block and are not saved in the active program.
To modify a programmed block use this procedure:

1. Select the BLK/BLK operative mode
2. Execute the program up to the block to be edited.
3. Select the MDI mode.
4. In the input window, press the "***Modify"*** pushbutton to move the program block on the data entry window.
5. Edit the block displayed in the data entry window and press "***Enter***" to confirm the modifications.
6. To execute the modified block press "***Cycle Start"***.
7. To quit the Block Modify select an operative mode different from MDI.

NOTE:
This procedure is not applicable when tool offset is active (G41, G42).

7.4 Strings search

The following procedure permits to search for a string of characters in an active program.

1. Open the data-entry "***Text Search***" using the "***Part Program***" – "***String Search***" softkey command
2. Insert in the input field the text to search
3. Press the "***UP***" button to start the search backwards. Press the "***DOWN***" button to start the search

When the search is completed, if one block contains the text required, it is displayed backwards on the screen. If the string is not found after searching in a given direction, one of the following error messages appears: "End of File" or "Beginning of File".

7.5 Feedrate Override

The feedrate value to be used in the program is specified by the F value parameter. To alter this value without modifying the program use this procedure:

1. Use the mouse to activate on the screen the button to select the overrides and select FRO (Feed Rate Override).

2. Using the buttons "+" and "-" on the right, increase or decrease the present override value. Each time one of the two button is pressed, the override percentage has a 1,25% increase/decrease. The allowed override has a range from 0% to 125% of the value configured, unless it has been programmed the maximum feed configures. In this case, the override has a range from 0% to 100%.

Example:
If the programmed feedrate value is 2000 MMPM (mm per minute) and a 50% variation is applied, the actual feedrate will be 1000 MMPM.

7.6 Spindle feedrate override

The procedure to change the spindle feedrate, programmed in the part program in execution with the "S value" operator is:

1. Use the mouse to activate on the screen the button to select the overrides and select SRO (Spindle Speed Override).

2. Using the buttons "+" and "-" on the right, increase or decrease the present override value. Each time one of the two button is pressed, the override percentage has a 1,25% increase/decrease. The spindle override percentage has a range from 50% to 150% of the programmed value.

Example:
If the programmed spindle feedrate is 1500 RPM (round per minute) and you select a 50% override, the real spindle feedrate will be 750 RPM.

7.7 Rapid Feedrate Override

The procedure to change the rapid feedrate configured in AMP is:

1. Use the mouse to activate on the screen the button to select the overrides and select RAP (RAPid override).

2. Using the buttons "+" and "-" on the right, increase or decrease the present override value. Each time one of the two button is pressed, the override percentage has a 1,25% increase/decrease. The rapid feedrate override has a range from 0% to 125% of the programmed value.

Example:
If the programmed rapid feedrate is 1500 3000 MMPM (millimetres per minute) and you select a 50% override, the real spindle feedrate will be 1500 MMPM.

7.8 Execution in *Dry Run* mode

In Dry Run mode, it is possible to try to execute a part program without any command being sent to the drives of the axes or to the machine logic (I/O inhibited). This feature is activated by pressing the "***Setup***" softkey and then "***Dry Run***".
With Dry Run active, it is also possible to verify program correctness graphically by way of the WinNBI Machine Plot utility.
For further details about use of Machine Plot, see Ch.11.

When "***Dry Run***" is active, the block reading speed is inversely proportional to the values selected for FRO and RAP.
The higher is the percentage, the slower is the block read rate. Maximum block read rate is equivalent to 0%.

7.9 Memory Search

Memory searching means searching for the interruption block and subsequently resume the interrupted cycle automatically or searching for a pre-set block where to start machining. The parameters which identify univocal an executing cycle are stored and continuously updated during the execution of the program. According to these parameters it is possible to carry out an automatic searching.

Memory searching can be accomplished in two ways:

- **Automatic memory search for the interruption block** (due to switch-off or reset of the machine)
- **Memory search for a pre-set block.**

7.9.1 Automatic Search

To search automatically for the block which interrupted the part program, proceed as follows:

1. Select the part program in which machining was interrupted.
2. Set "***Automatic***" operative mode
3. Select the "***Setup***" softkey and then "**Search in memory**": the system starts searching. The screen displays the reverse **MemS** flag.
4. Press "***Cycle Start"***: the system runs the searching and in the end the message **NC156 end of search in memory** and the programmed dimensions of the axes position for the cycle reset are displayed. The block of the part program which will be executed at the end of the search, when the program is restarted, is displayed on reverse image. By switching to BLK/BLK and pressing "***Cycle Start"***, the search can be continued to the desired block.
5. Select again the softkey "***Setup***" and then "***Search in memory***" to exit the status of the stored search.

The system is now ready to restart the working cycle.

7.9.2 Search for a pre-set block

To search for a Nxxxx block of a part program proceed as follows:

1. Select the part program
2. Set automatic mode
3. Select the "***Setup***" softkey and then "***Search in memory***": the system starts searching. The screen displays the reverse **MemS** flag.
4. Select the "***Part Program"*** softkey and then "***Exec From To***", set the starting and the finishing blocks for the memory searching in the data entry. The starting block can also be omitted (by default part program starts with the first block); the finishing block stands for the previous block from which you want to restart the cycle. The searching blocks cannot be included in a subprogram
5. Press "***Cycle Start"***: the system runs the searching and in the end the message **NC 156 End of search in memory** and the programmed dimensions for the position of the cycle reset are displayed. The block of the part program which will be executed at the end of the search, when the program is restarted, is displayed on reverse image. By switching to **BLK/BLK** and pressing "***Cycle Start"***, the search can be continued to the desired block.
6. Select again the "***Setup***" softkey and then "***Search in memory***" to exit the search memory status. on the scree the flag **MemS** turns off.

Before exiting the search status, it is possible to continue the searching up to another block resetting the number by means of the "***Exec From To"*** softkey and pressing "***Cycle Start"*** again.

7.10 Restart a working cycle

After exiting the MEMORY SEARCH mode to restart the working cycle, proceed as follows:

1. Press "***Cycle Start***": the system send auxiliary functions to the machine logic and enters HOLD status. Auxiliary functions are sent also in SEARCHING mode provided that the logic has been adequately pre-set.
2. Position axes on the programmed dimensions reversing on the profile (**JOG RETURN** softkey, **MANUAL** mode).
3. Select again automatic or semiautomatic mode.
4. Press the button "***Cycle Stop***" to exit from **HOLD** status.
5. Press "***Cycle Start*** "to restore the working cycle.

7.11 Searching Mode

- When using multiprocess systems, memory searching can be accomplished on the first 4 processes only.

- When an interrupted cycle is restored automatically, the program is restarted from the beginning of the block where the interruption took place. When a cycle is restored from a pre-set block, it is restarted from the block next to the one searched.

- Automatically resuming the search, if the position of the start of the search is changed, for example, switching to **BLK/BLK** and using the arrow keys or setting the starting block only in the data entry ***"Exec From To"***, the search starts from the new position and ends when the system has executed the number of blocks stored throughout the machining phase.
  The same thing happens when a search is carried out for a set block if the end block is not specified.

- Automatic searching is only possible provided the relevant configuration (AMP) is enabled. It can only be used when machining is executed at the beginning of the part program. Similarly, if during a working cycle MDI-blocks are executed, memory searching subsequently run on that part program would not work.

- When machine is switched off, modified data-entry from ***"Program Set Up"*** or from other data entries are erased.

- When searching, variable allocation blocks are always executed.

- Searching is not operating properly when part program makes use of variables written or read from logic or from other environment.

- Following are listed the strings that can and cannot be used within a part program in order to execute a memory searching:

  The following strings are accepted in searching mode:
  DAN, IPB, UAO, UTO, UIO, SOL, DPA, PAE, PAD, MIR, ROT, SCF, AXO, LOA, RPT, ERP, CLS, PTH, EPP, EPB, GTO, IF, ELSE, ENDIF, GDV, RDV, UGS, CGS, DGS, DIS, DSB, GTA, ECM, PRO, PVS, GTP, CCP, SPA, SPF, SPP, CLP, TGL, OPN, WRT, REA, CLO, DEL, INS, PRO, RTP, ROP, GPS, PLS.

The following string is accepted but non-executed in searching mode:
DLY.

The following strings are not accepted in searching mode:
RQO, RQT, RQP, TOU, DPP, UDA, SDA, UPR, UVP, UVC, TCP, REL, WOS, SND, WAI, EXE, DCC, FIL, SOP, GET, PUT, SCL, EPS, CON, COF, HON, HOF, RES, SMD, SAX, DIR, JOG, FHO, UVA,

Besides the following tracing cycles are not accepted: G72, G73, G74.

- Block reading speed is inversely proportional to the values selected for FRO and RAP. The 0% corresponds to the maximum block reading feed.

8. Using Tables

The system manages several groups of parameters which, if they were not structured in an organised way, would be difficult to manage. These parameters have been arranged into a number of tables. These tables are:

- Origins
- Tools
- Tool Offsets
- User

In the CNC OPENcontrol system, the tables are stored in a dual port memory, accessible directly from the machine logic (WinPLUS) or by the ISO user program. The tables are saved even after the system has been turned off.

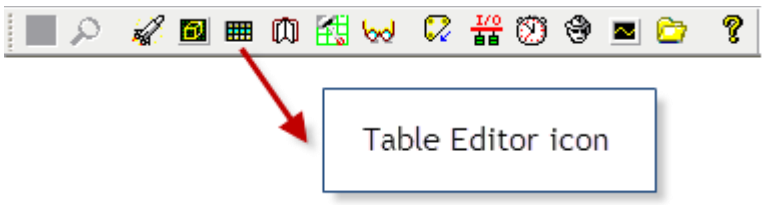
Each of these tables is described later in this chapter. The tables are managed by the Table Editor, allowing both the tables stored in the dual port memory and the tables stored on the hard disk to be changed. Here are some of the editing operations available with the Table Editor:

- Table uploading, opening and saving
- Modification of table data
- Incremental modifications to table fields
- Measure unit conversion (mm/inch)
- Backup and restore of the tables or all of the CNC memory

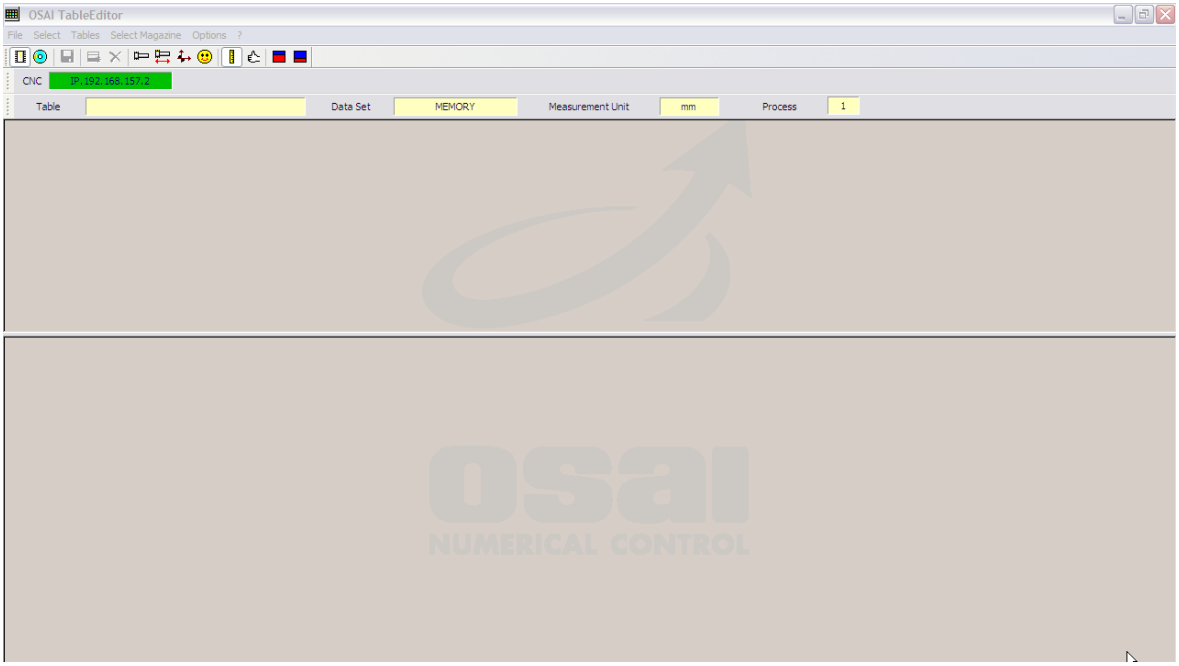
This manual describes the standard table configuration. The table structure can be modified by the OEM (amount of data, name and position of the fields) and can then appear to the end user in a manner different from the standard: in that case the OEM should describe the table functions.

8.1 Enabling Table Editor

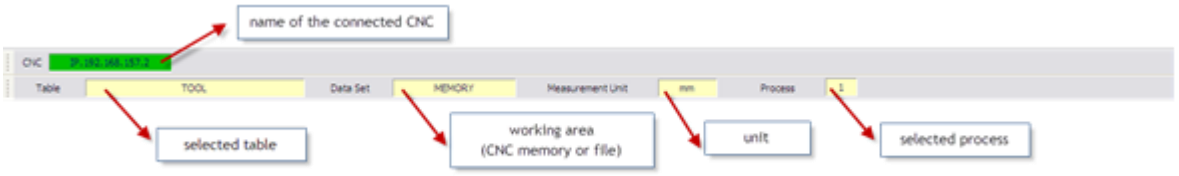
To enable the Table Editor, just press the button in the icons bar



or select UTILITY and then TABLE EDITOR from the Tool Bar. At this point you will see the following window.

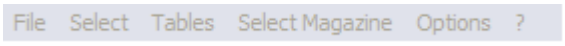


The upper section displays the following information:

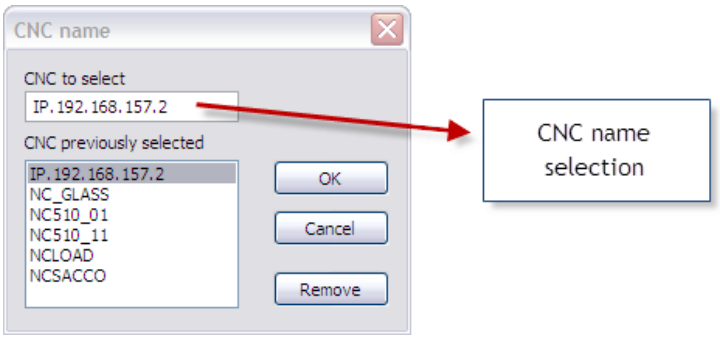


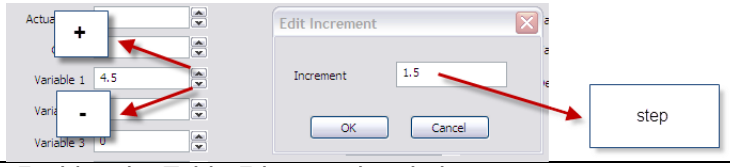
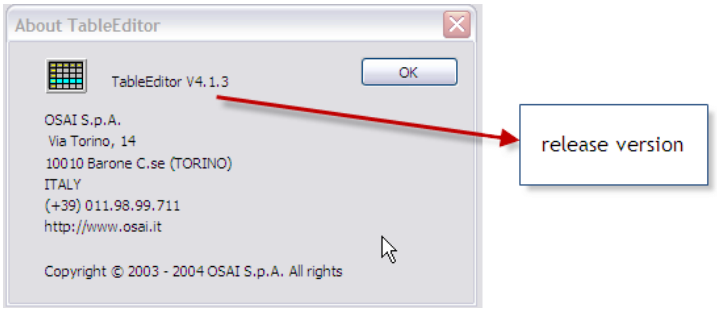
- name of the connected CNC
- selected table (TOOL, OFFSET, ORIGIN, USER)
- working area (CNC memory or file on PC)
- unit of measurement (mm or inch)
- selected process

The following commands are available in the upper bar:

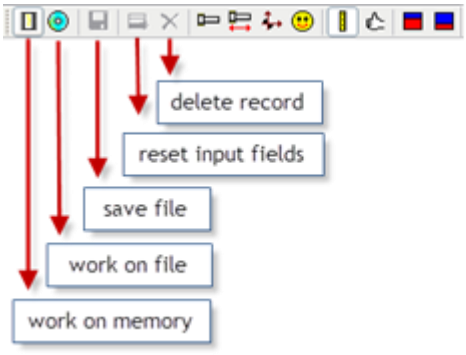


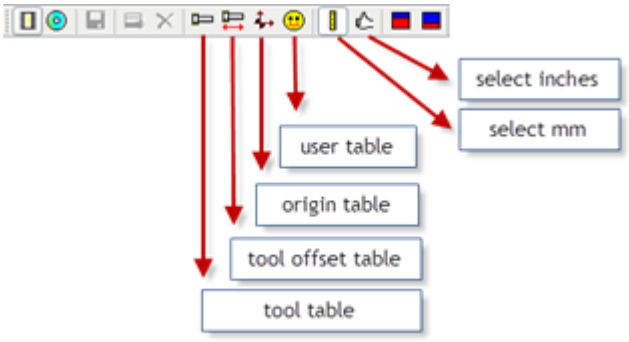
FILE	Table Builder	This command allows you to access the Table Builder that is described in the next chapter. This ambient is reserved for the OEM, therefore it is password protected. Using this tool, the tables can be customised, i.e. configuring which tables and fields are displayed to the end user. It also configuration of up to 16 customised tables, where each field has a custom label. Finally, the graphical representation, i.e. the field positions can be customised, the text alignment, the dependence of the selected measurement units, etc.
	Save	This command is only available when working on files. It allows the changes made to the data in the file to be saved to the file.
	Save As	After making changes to a file, using this command the changes can be saved to a file with a different name. If working with the CNC memory, by using this command the data can be saved from the CNC memory to a file.
	Update Memory	With this command data can be copied from a file to the corresponding table in the CNC memory: this is the reverse command to “Save As”.
	Backup Memory	This command allows the entire area data of the tables (tool, offset, origin, user) to be saved from the CNC memory to a single file.
	Restore Memory	This command allows all the data relating to the data area (tool, offset, origin, and user) to be copied from a file to the memory of the CNC. This is the reverse command to “Backup Memory”
	Work on Memory	Allows the memory of the selected CNC to be modified. This is possible only with a connected CNC. Any change made is immediately active on the CNC and will affect its operation.
	Work on File	Allows a file to be modified, therefore the changes made do not affect the CNC functioning. With this tool it is possible to prepare data files relating to the tools and to the origin without being connected to a CNC.
	Print	Command unavailable.
	Exit	Exit from Table Editor.

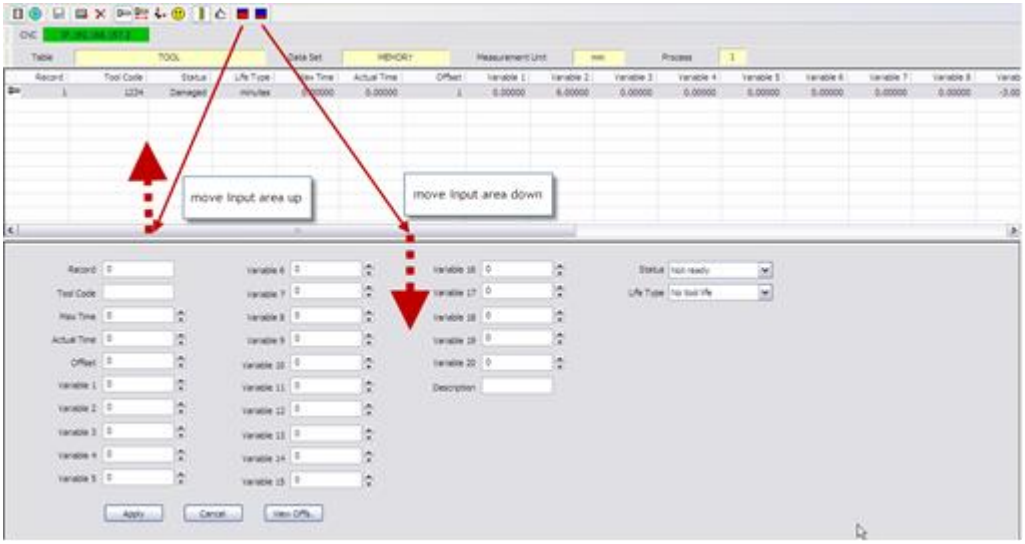
SELECT	CNC	This command defines the CNC name with which the Table Editor will communicate. This setting is stored when the Table Editor is exited and the connection with the selected CNC is automatically restored following the next reboot of the Table Editor. 
	Process	This command defines the process number with which the Table Editor is associated. This is effective only when the Table origins change: because, the Tool, Offset and User tables apply to all processes while the Origins are specific to each process.
	Project	This command allows a “project” created with the Table Builder to be opened and changed and saved. Therefore, this is a command reserved for the OEM.
TABLES	Tool	If the “Work on Memory” mode is selected, this command allows the tools table to be changed directly in the CNC working memory. If the “Work on File” mode is selected, the name of the tools data file to be edited is required.
	Offset	If the “Work on Memory” mode is selected, this command allows the tool offset tables to be changed directly in the CNC working memory. If the “Work on File” mode is selected, the name of the tool offsets data file to be edited is required.
	Origin	If the “Work on Memory” mode is selected, this command allows the origins tables to be changed directly in the CNC working memory. It should be noted that the origins are specific to each process. If the “Work on File” mode is selected, the name of the origins data file to be edited is required.
	User	If the “Work on Memory” mode is selected, this command allows the user tables to be changed directly in the CNC working memory. If the “Work on File” file, the name of the user data file to be edited is required.

SELECT MAGAZINE	All 1..10	Selects the magazine to which the Tools Table applies (optional). Unavailable command.
OPTION	Edit Increment	This command allows definition of a value that will be added or subtracted from the value of the selected field in the table for each press of the button 
?	Help	Enables the Table Editor on-line help
	About Table Editor	Using this command displays the Table Editor release version. 

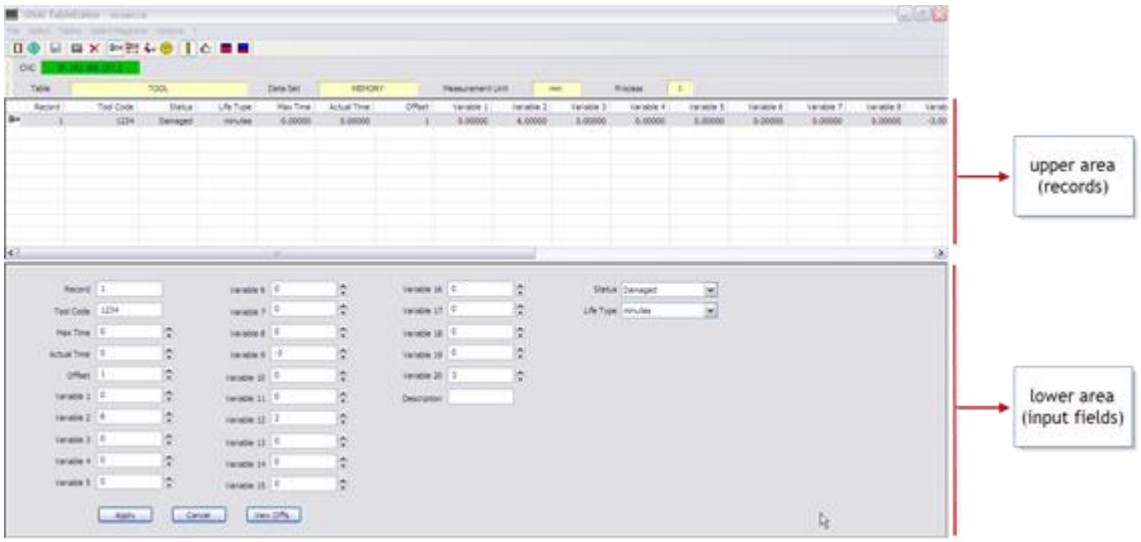
Some of the commands described above can be accessed directly from the Tool Bar, as outlined in the following figures:







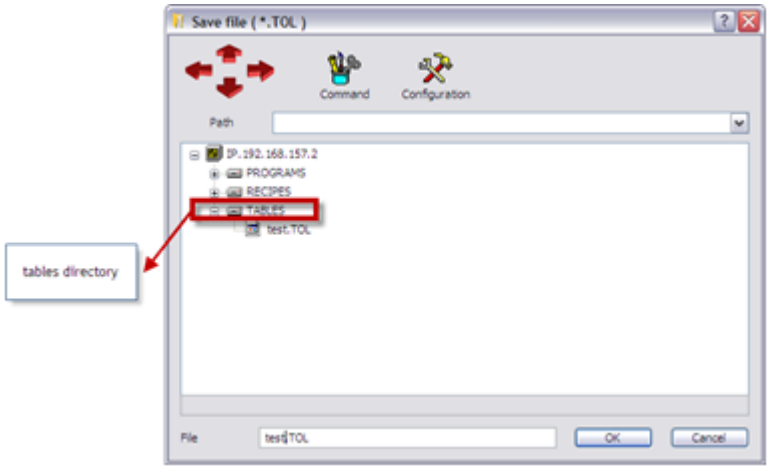
When a table is displayed (whether as a file or the memory of the CNC) data are displayed and modified in the same way.
The window is divided horizontally into two parts: at the top of the table, data are displayed in rows (records) and columns (fields). When a row is selected at the top, all the corresponding fields are displayed at the bottom where they can be edited and modified.



A field can be changed directly by writing a value or by using the increment/decrement buttons on the side or (in some cases) by selecting a value by using a drop-down menu. At the end the APPLY button can be pressed to save the changes, or CANCEL to restore the original value.

If working on the MEMORY, using “APPLY” saves data to the CNC working memory and it is immediately active. If working on the FILES, using “APPLY” saves the data entered in the bottom window into the matching record. All the data is permanently stored in a file only by using the command SAVE or SAVE AS.

The files are saved by default in the TABLES folder on the CNC hard-disk, but it is also possible to select and to use a different folder (e.g. PC hard disk).



A name may be inserted for each file but the extension is automatically set according to the following rule:

- TOL : tools file
- OFS : offsets file
- ORG : origins file
- USR : user file
- IMA : backup file of the entire data area

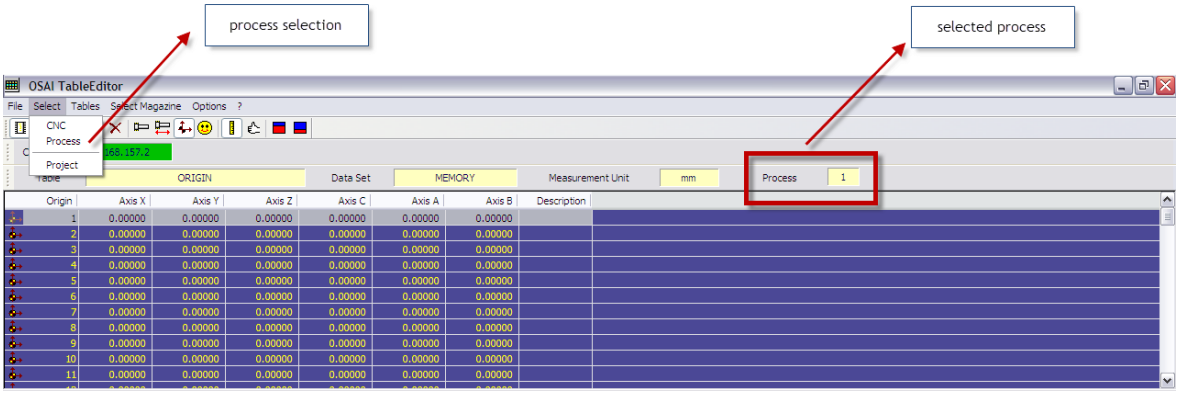
These files can also be copied to a different CNC where they can be installed in the memory of the new CNC via the UPDATE and RESTORE MEMORY commands.

8.2 Origin Table

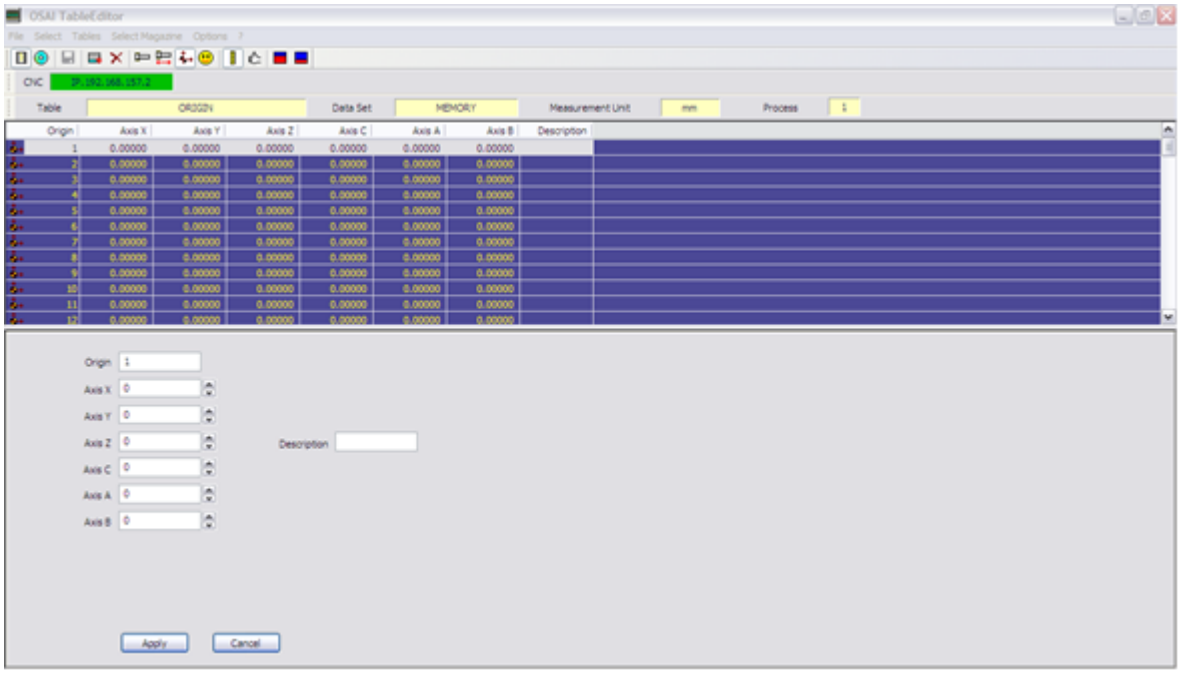
While all the other tables are common to all processes, Tables of Origins are specific to the process. Up to a hundred origins (each for up to 12 axes) for each of the processes configured can be defined.

When there is more than one process, it is very important to pay attention to which process has been selected to avoid to changing the origins of the wrong process.

Process selection is done by pressing SELECT on the tool bar and then PROCESS. The currently selected process is highlighted in the information area above the table.



When Origin Table is selected , the window below appears



where the fields have the following meaning:

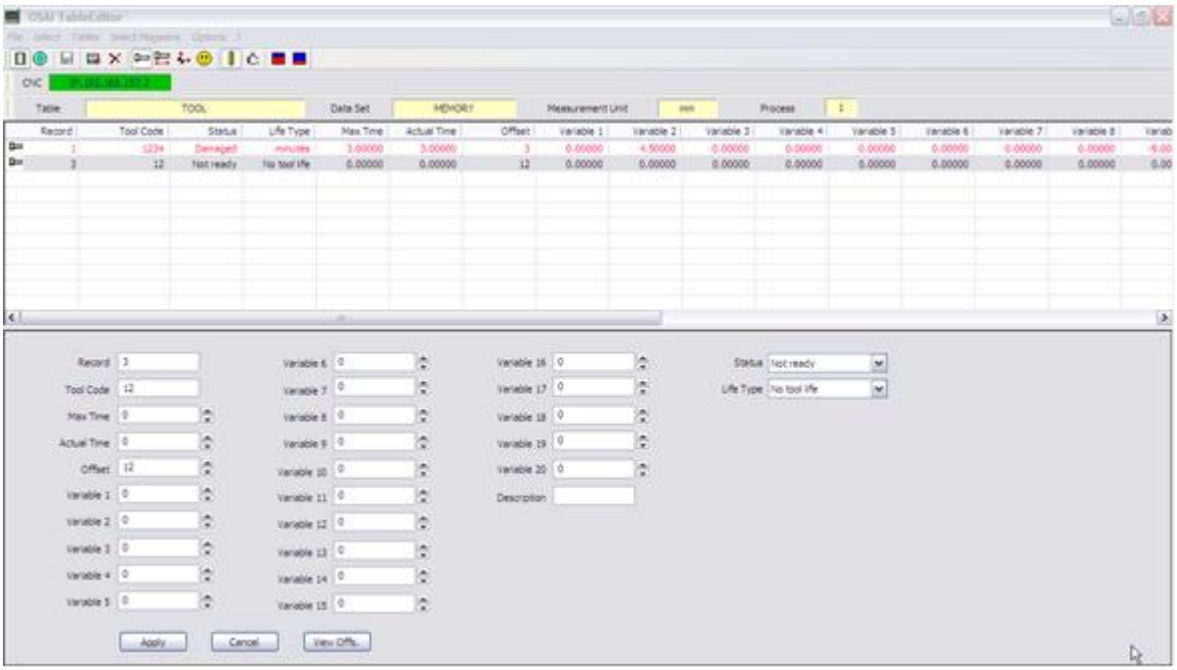
Origin Is the origin number that can have a value between 1 and 100 (There are 100 origins available).

Axis … It is the origin related to the indicated axis (up to 12 axes).

Description It is a character type field where a comment may be inserted to describe the origin.

8.3 Tool Table

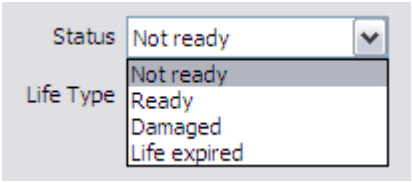
When the Tool Table is selected, the following window appears



where the fields have the following meaning:

Record	The record number that can have a value between 1 and 250 (up to 250 tools can be configured)
Tool code	Up to 32 alphanumeric characters can be used to identify the tool.
Max time **Actual time**	These two fields define the maximum and the actual tool life. The values to be inserted depend on the life type set using the LIFE TYPE field. The management of these areas is the responsibility of the machine logic developed by the OEM and works only if provided for in the application.
Offset	It is the offset number associated by default to the tool. It is a number between 1 and 300. Up to 300 offsets can be configured.
Variable 1..10	These are decimal numerical variables whose meaning is defined by the OEM, therefore it depends on the application.
Variable 11..20	These are integer numerical variables whose meaning is defined by the OEM, therefore it depends on the application.
Description	It is a character-type field where a comment to describe the tool may be entered.
Status	It is the tool status. Possible values are those that can be selected using the drop-down menu: Not ready: tool unavailable Ready: tool available

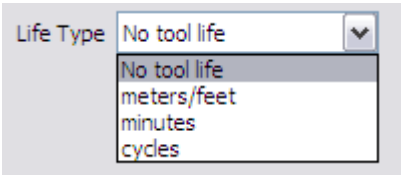
Damaged: tool damaged
Life expired: tool life expired



The management of this field is controlled by the machine logic developed by the OEM and works only if managed by the application

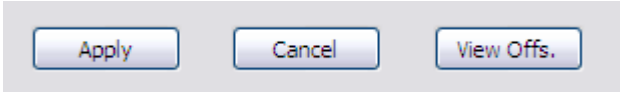
Life Type

It is the tool life type. Possible values are selected by using the drop-down menu:
No tool life: life tool management disabled
Metres/feet: tool life expressed by the distance travelled by the tool
Minutes: tool life expressed in operating time of the tool
Cycles: tool life expressed in work cycles

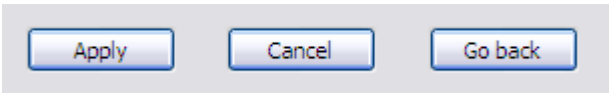


The selection of the life type determines the type of values to be entered in the fields MAX TIME and ACTUAL TIME.
The management of this field is controlled by the machine logic developed by the OEM and works only if provided for in the application.

After entering the necessary changes to the fields they can be stored via the APPLY button or aborted using the CANCEL button.

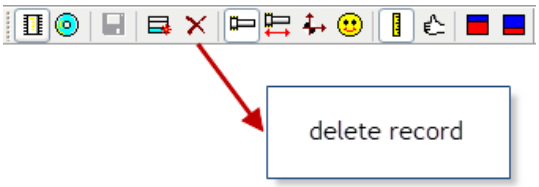


The VIEW OFFS button allows access to the offsets Editor, and the offset associated with the selected tool. The data of the associated offset can be modified and then the Tool Editor reselected by pressing GO BACK.



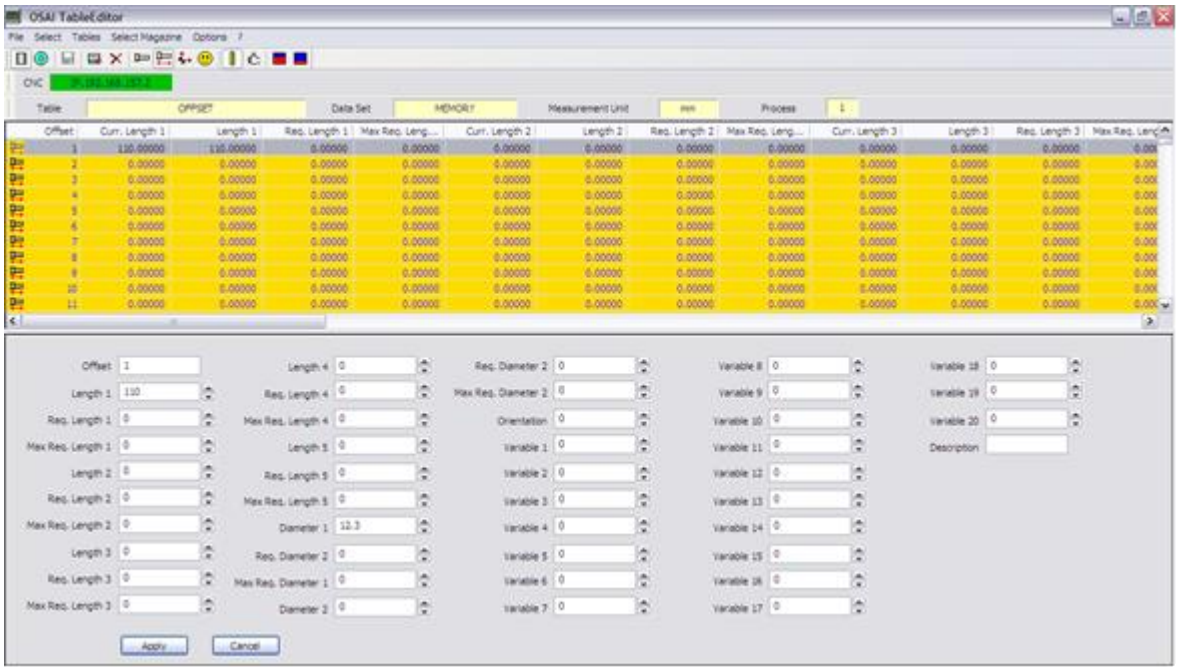
In order to insert a new table tool, simply enter in the RECORD field an unused record number, then the appropriate values in the following fields. When the APPLY button is pressed, a confirmation is requested by a message: by responding OK, a new record is inserted in the table.

To delete a tool from the table, simply select the corresponding record and then press the delete button: after a further confirmation of the record is deleted.



8.4 Tool Offset Table

When the Tool Offset Table is selected, the following window appears



where the fields have the following meanings:

Offset The offset number can have a value between 1 and 300 (there are 300 offsets available).

Length 1..5 This is the nominal tool length value.
Usually, for milling machines, only one tool length is used (typically applied along the Z axis) while for lathes two lengths are used (applied along X and Z), but the OPEN*control* CNC can define and apply up to 5 lengths on 5 different axes.

Req. Length 1..5 This is the current re-qualification value applied to the nominal Length 1..5 parameter.
This value can be changed manually or automatically updated by the machine logic after a tool calibration cycle. The management of this field is then managed by the machine logic developed by the OEM and works only if provided for by the application.

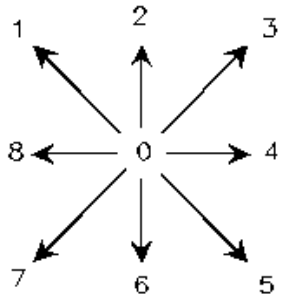
Max Req. Length 1..5 This is the maximum possible re-qualification value that can be applied. If set to 0 no limit exists. When this value is reached the tool is considered “expired”: this management is handled by the machine logic developed by the OEM and works only if required by the application.

Diameter 1..2 This is the nominal tool diameter value.
Typically there is only one tool diameter defined, but the OPEN*control* CNC can define and apply up to a maximum of 2 different diameters. Such management, however, is managed by the machine logic developed by the OEM and works only if provided by the application.

Req. Diameter 1..2 This is the current re-qualification value applied to the nominal diameter 1..2 parameter.
This value can be changed manually or can be automatically updated by the machine logic after a tool calibration cycle. The management of this field is then handled by the machine logic developed by the OEM and works only if provided for in the application.

Max Req. Diameter 1..2 This is the maximum possible re-qualification value that can be applied. If set to 0 no limit exists. When this value is reached the tool is considered “expired”: this management is handled by the machine logic developed by the OEM and works only if required by the application.

Orientation This is the orientation of the tool tip with respect to the interpolation plane. It can vary from 0 to 8 as shown in the figure below:



Values from 1 to 8 are used for lathes and grinding machines. For milling machines, the value is always 0.

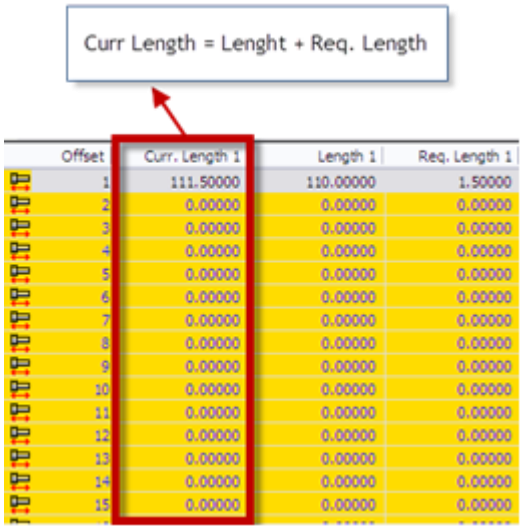
Variable 1..10 These are decimal numerical variables whose meanings are defined by the OEM and therefore depend on the application.

Variable 11..20 These are integer numerical variables whose meanings are defined by the OEM and therefore depend on the application.

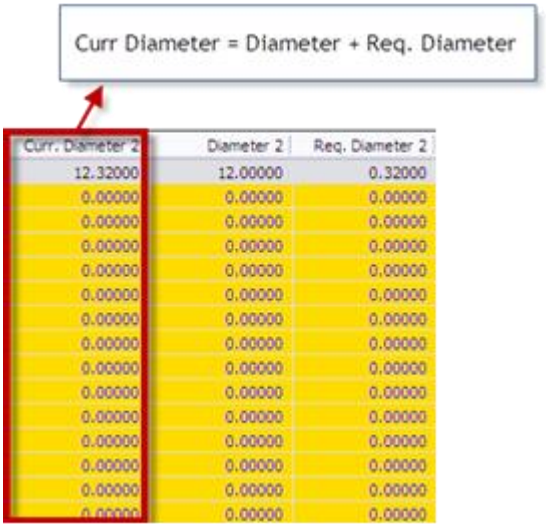
Description It is a character type field where a comment to describe the tool may be inserted

Note that in the upper part of the screen, more fields (columns) are displayed. than those in the lower data insert area. These fields are:

- CURR. LENGTH1..5 (actual tool length): this field is automatically updated from the system as a sum of the nominal length value (LENGTH1..5) and its requalification (REQ. LENGTH1..5)

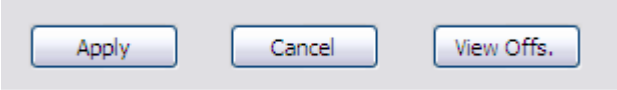


- CURR. DIAMETER1..2 (actual tool diameter): this field is automatically updated from the system as a sum of the nominal diameter value (DIAMETER1..2) and its requalification (REQ. DIAMETER1..2)

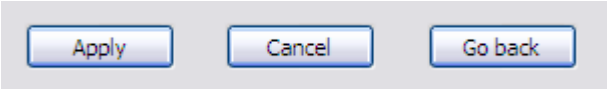


After entering the changes to the fields store them via the APPLY button or abort using the CANCEL button.

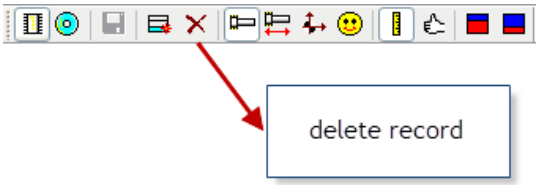
If the Tool Offset Editor is enabled by the Tool Editor pressing the button VIEW OFFS



in addition to the two buttons APPLY and CANCEL the GO BACK button is also displayed which permits return to the Tool Editor

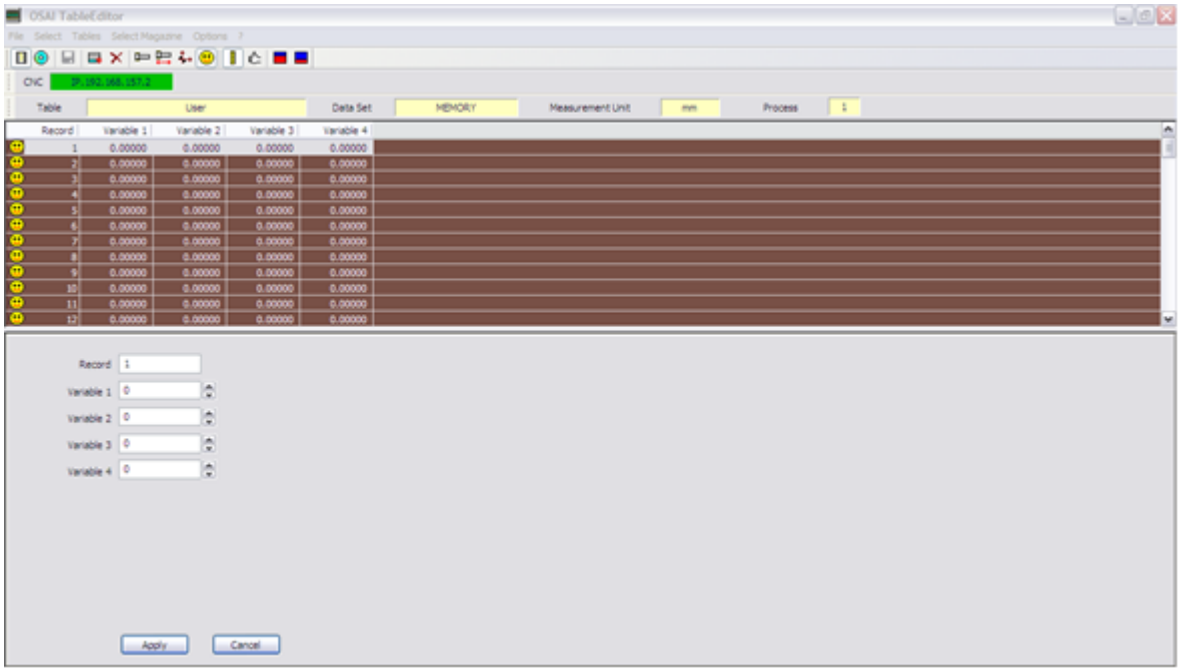


To delete a tool offset select it and then press the delete button: after a further confirmation of all fields of the offset will be deleted.



8.5 User Table

The User table has 100 records, each of them with 4 variables, whose meaning is defined by the OEM, therefore it depends on the application.
When the User Table is selected, the window below appears:



where the fields have the following meaning:

Record The number of the record that can have a value between 1 and 100

Variable 1..4 These are the 4 numerical decimal variables referring to the selected record. Their meaning is defined by the OEM and therefore depends on the application.

After entering the necessary changes to the fields store them via the APPLY button or abort using the CANCEL button.

Please note that this ambient can be heavily customized by the OEM and may then appear completely different to the end user.

9. The Table Builder

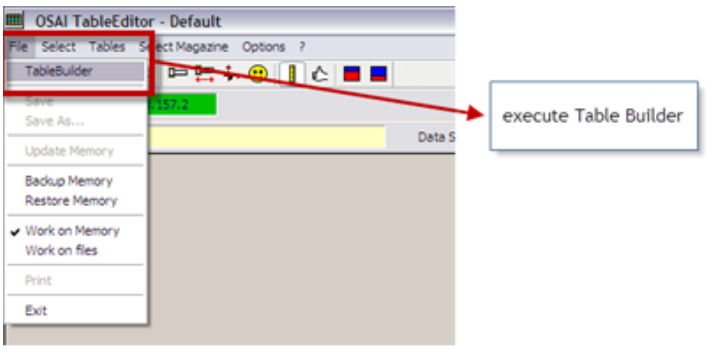
9.1 General

The Table ambient described in the previous chapter, can be customized using the Table Builder to satisfy specific users' requirements. This operation is usually done by the OEM during the application development, but changes can be inserted at any time.

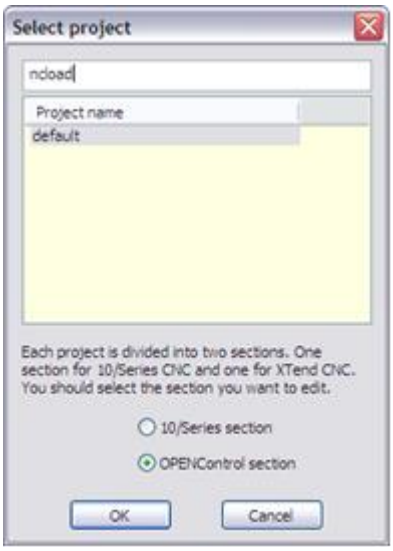
The configuration and any subsequent changes have to be made by qualified staff, having sufficient system knowledge to evaluate the possible consequences that these operations (configuration of system changes) can have on the system. For this reason access to this area is protected by a password. Using this tool it is possible to:

- Define different "projects", that is general table configurations or particular configurations customized according to the connected CNC
- define which tables the end user can view
- define which are the fields that can be viewed and which can be modified by the end user
- define up to 16 different user tables, whose associated record numbers, parameter names and position in the display can be customized

The Table Builder is enabled by selecting FILE and then TABLE BUILDER from the Tool Bar.



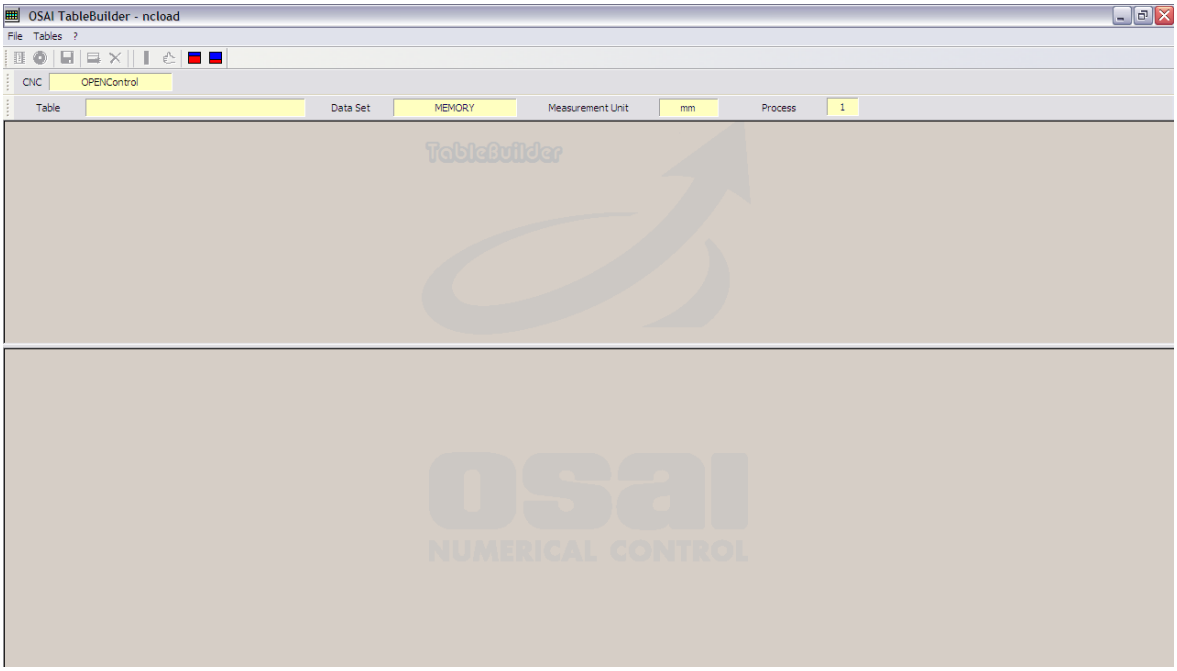
An access password is required: when the password is inserted, the window below appears. In this window the name of the project to be created or modified must be selected.


Select project
ncload
Project name
default
Each project is divided into two sections. One section for 10/Series CNC and one for XTend CNC. You should select the section you want to edit.
10/Series section
OPENControl section
OK
Cancel

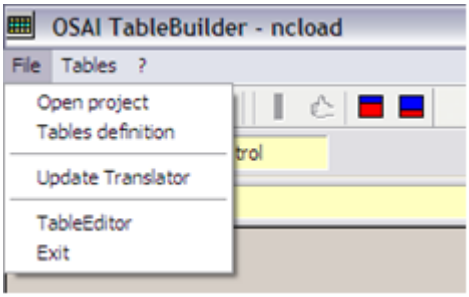
To create a new project, a new name that does not exist in the PROJECT NAME window must be entered. To change an existing project, select the name from the list in the PROJECT NAME window.

If the project has the same name as a CNC connected to the network, this project will be enabled when connected to the CNC having that name. This allows different projects to be associated with the machine to which the PC is connected. Alternatively, to connect to any CNC (independently of the name with which it is defined in the network) a project should be created having name DEFAULT. For instance, with reference to the previous image where two different projects are defined (NCLOAD and DEFAULT), if connected to a CNC named NCLOAD on the network, the project named NCLOAD will be automatically enabled. When connecting to a CNC with a different name, the DEFAULT project will be activated.

Always select the OPEN*control* SECTION option and then confirm with the OK button: at this point the project is opened and the following window is displayed.

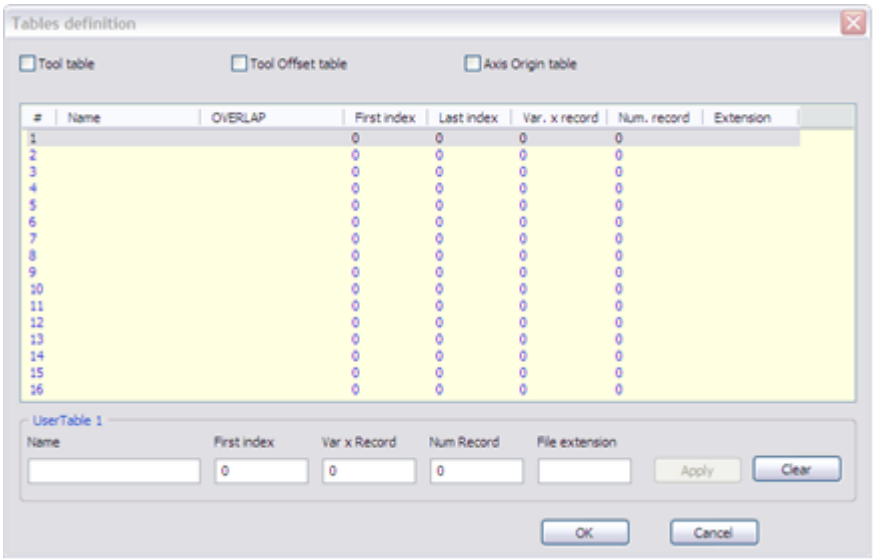


In the upper Tool Bar the following commands are available



FILE	Open project	This command allows the selection of a project
	Tables definition	This command enables the panel for configuration of the tables
	Update translator	This command creates or updates the .CSV file containing all the customized entries used in the tables and divided in columns by language so as to manage the language change using the WinNBI Config.
	Table editor	This command closes the Table Builder and returns to the Table Editor
	Exit	This command closes the Table Builder and the Table Editor.

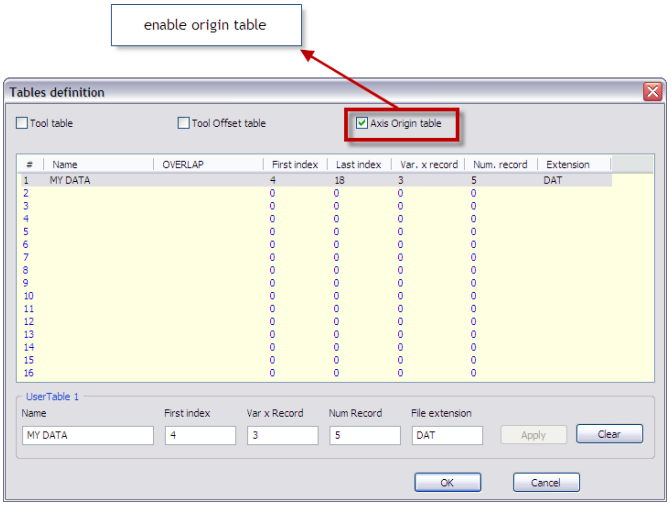
When TABLES DEFINITION is selected, the panel for the table configuration is displayed as below:



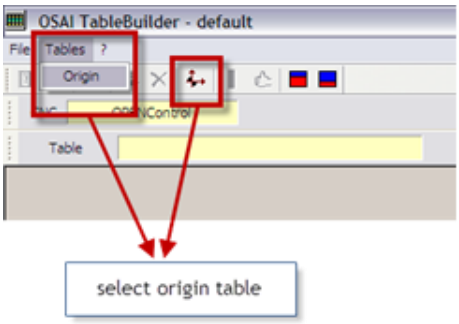
In the upper section of the window, there are three “check” boxes allowing enable/disable of the final user display corresponding to the table. In the lower section it is possible to define up to 16 customized user tables.

9.2 Origin table customization

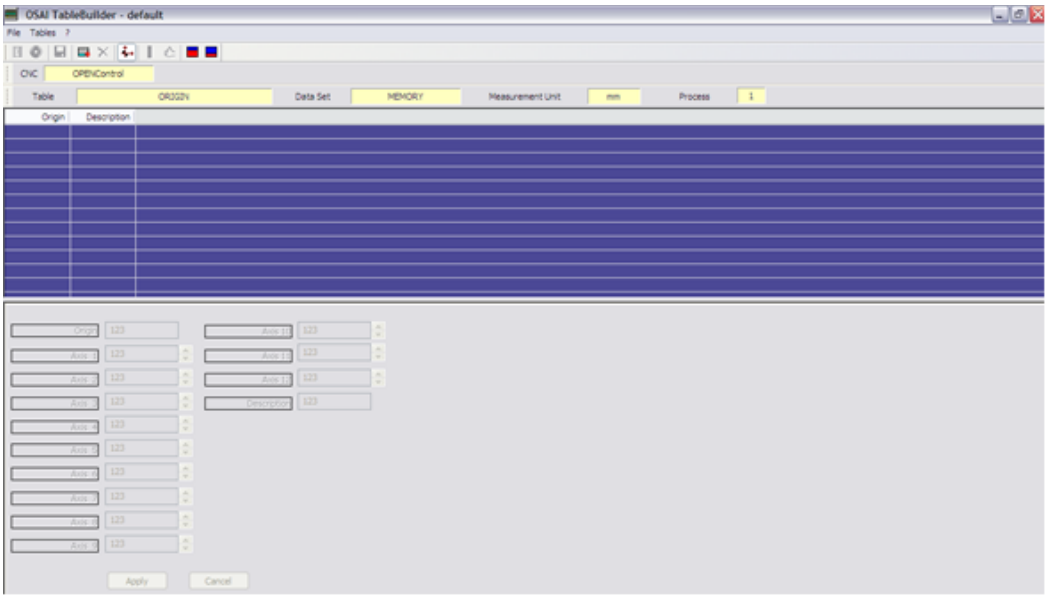
In the upper section of the TABLES DEFINITION window, it is possible to enable/disable the ORIGIN TABLE view.


enable origin table
Tables definition
Tool table
Tool Offset table
Axis Origin table
Name OVERLAP First index Last index Var. x record Num. record Extension
1 MY DATA 4 18 3 5 DAT
UserTable 1
Name First index Var x Record Num Record File extension
MY DATA 4 3 5 DAT
Apply
Clear
OK
Cancel

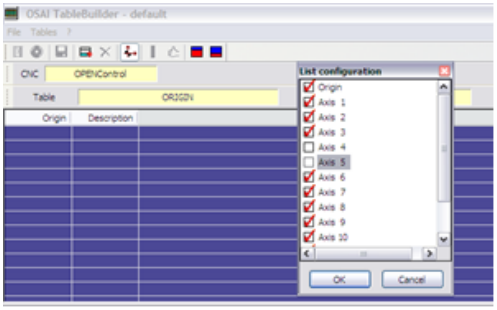
After confirming by pushing the "OK" button in the Tool Bar or using the dedicated button, it is possible to select the Origin Table



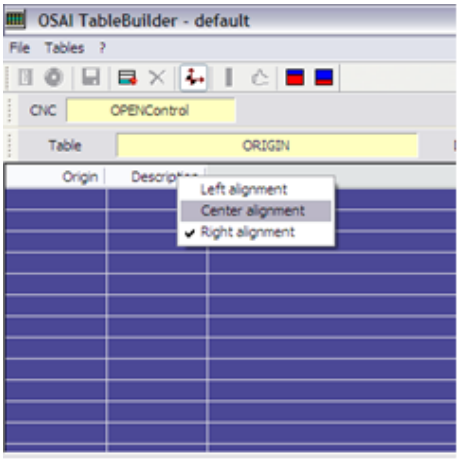
displayed as follow



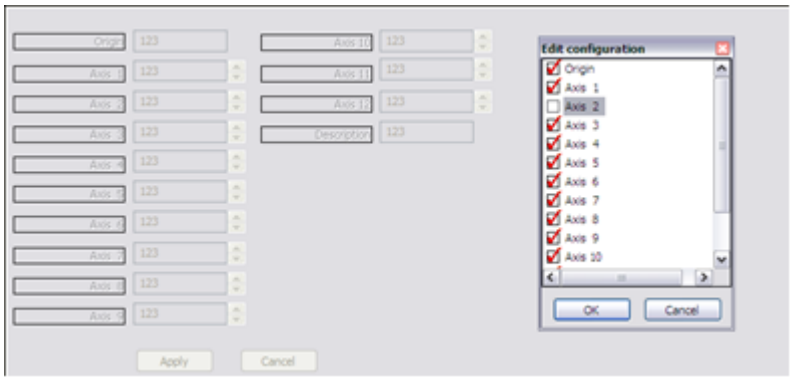
In order to change the view of the fields in the table above, click on the table with the right mouse button. A window is displayed where, by using a “check” box, it is possible to enable/disable the single column view. For instance, in the image below, the display of the origins referred to axes 4 and 5, is disabled.



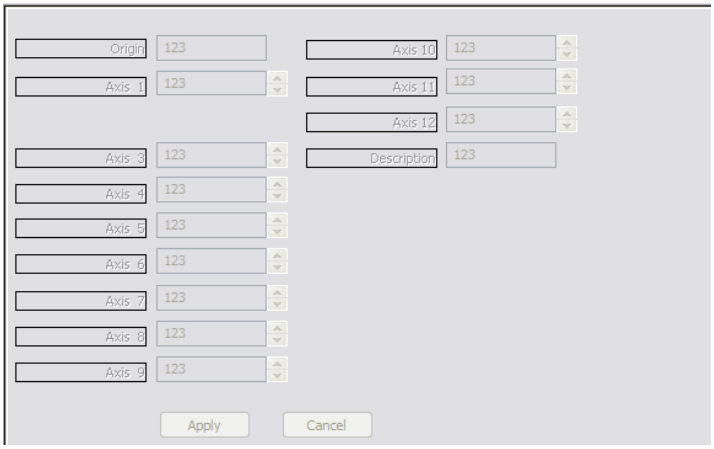
It is possible to change the field alignment for each column: for this reason, it is enough to right-click the column heading and to set the preferred choice. In the image below, for instance, the alignment of the second column is changed (DESCRIPTION) from RIGHT to CENTRE.



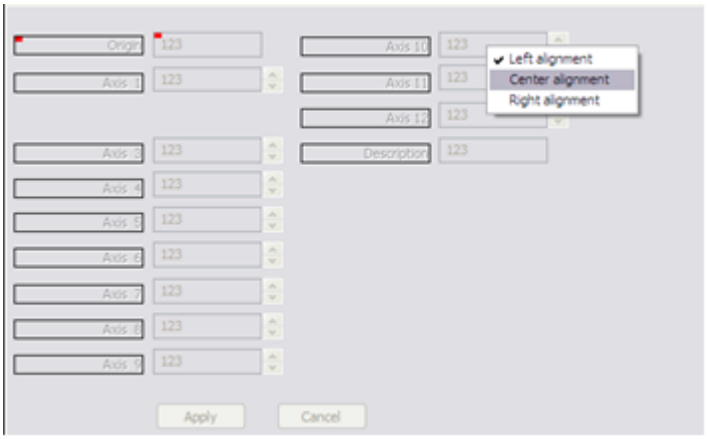
By right-clicking any point of the lower data insert window, it is possible to define which fields must be displayed and modified by the end user. For instance, in the picture below the origin of axis 2 is made invisible by clicking the related “check” box.



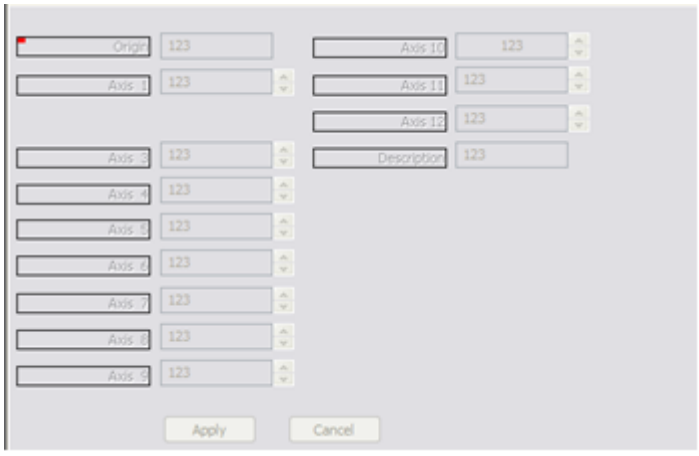
After the confirmation with “OK”, the axis 2 origin disappears from the screen.



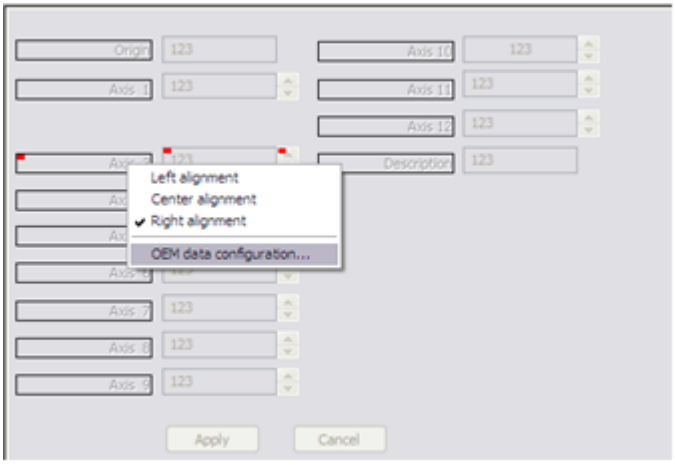
It is also possible to define the alignment of each field in the lower insert data window. It is enough to right-click on the field and then select. For instance, in the image below the user can change the alignment of the axis 10 origin from LEFT to CENTRE.



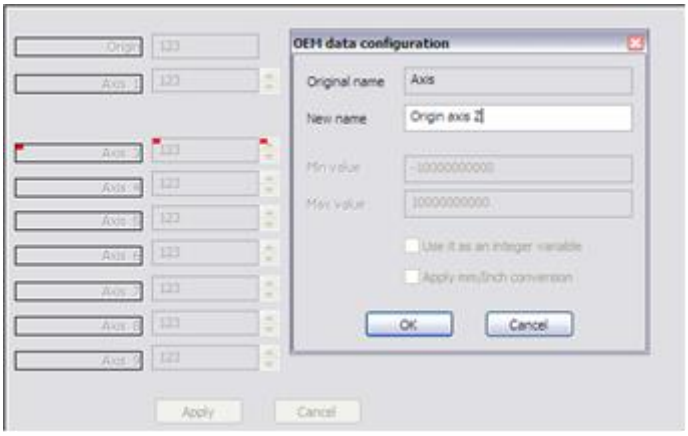
After the change, the field appears as the image below.



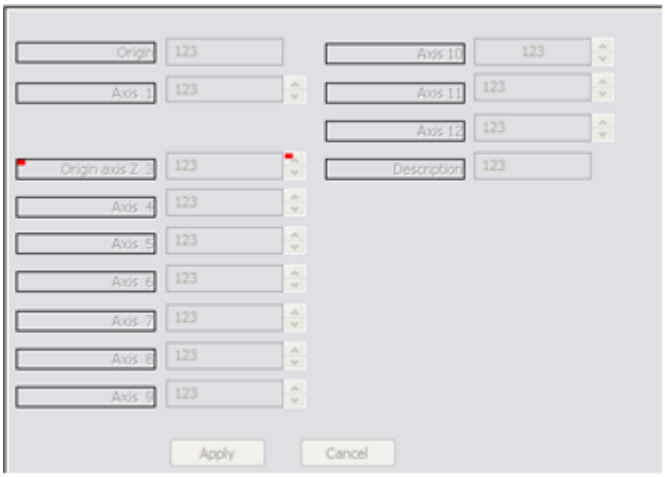
Another possibility is to define some priorities for each field. This can be done with right-click on the field label.



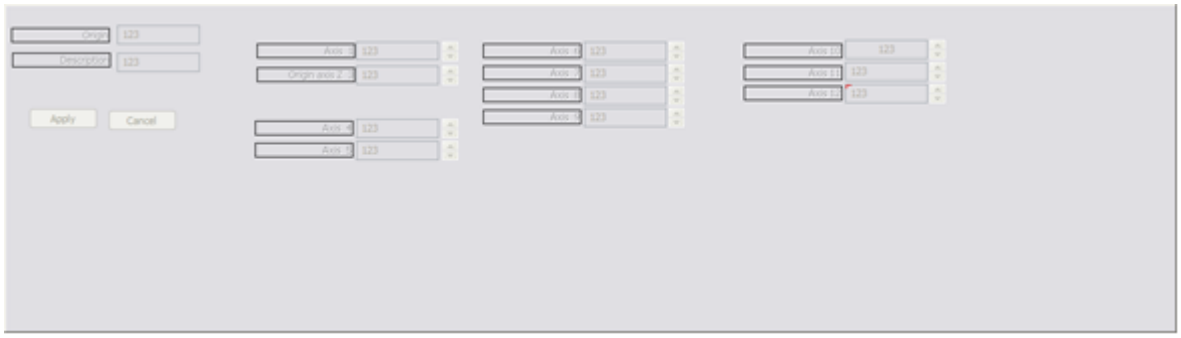
The MTB DATA CONFIGURATION window is opened and it is possible to change the variable name (in the case of origins, only the name can be changed).
For instance, in the image below, to change the name of the label referred to the axis 3 origin.



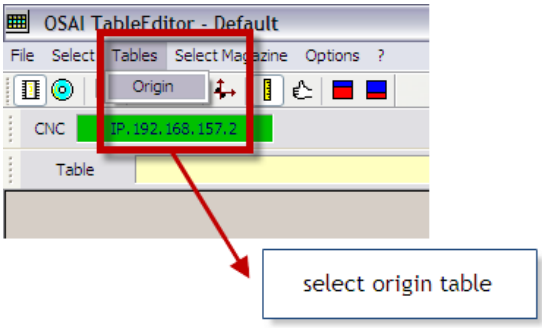
After confirming with the OK button, the variable will be displayed as below.



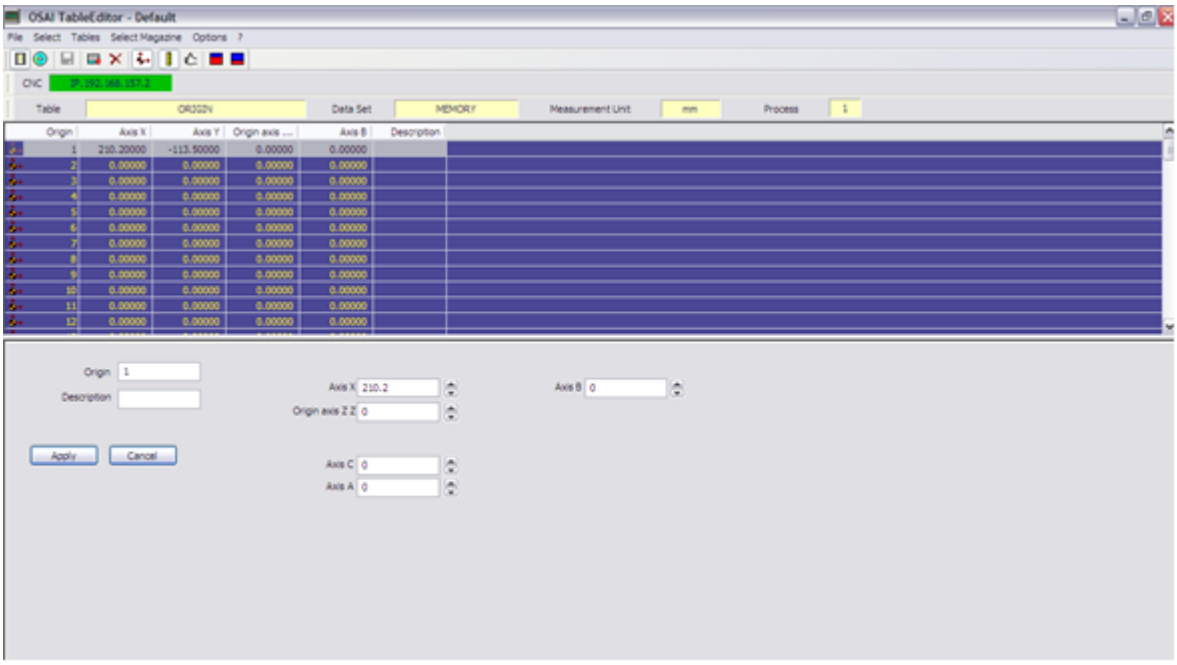
Using drag & drop, the fields in the lower window can be placed in different positions. When a field connected to another is selected (for instance label, value and spin-button), the three elements are outlined by a small red square. In the image below, the fields have been placed in a different layout.



The final result after the changes can be viewed from the Table Editor by selecting the Origin Table from the Tool Bar or from the appropriate button

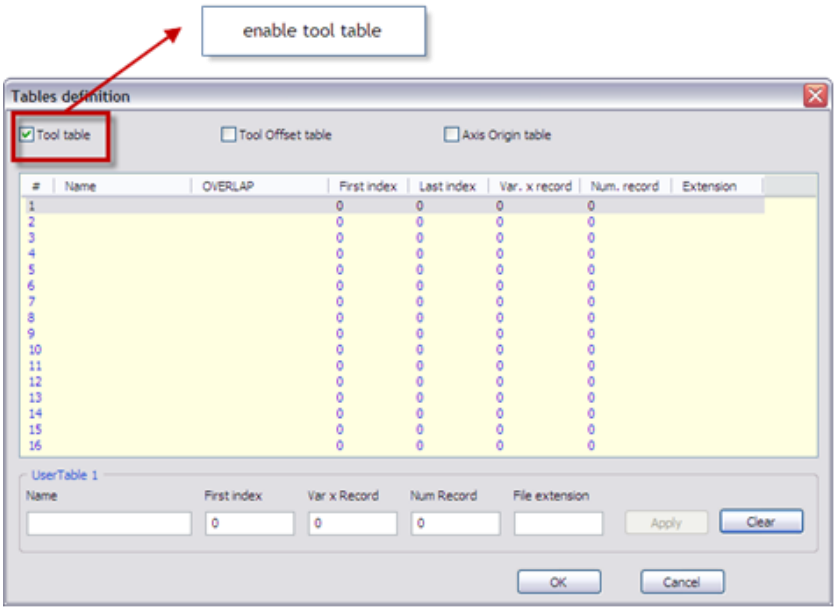


the following window only displays the axes configured in AMP and enabled for display according to the configuration created in this example.

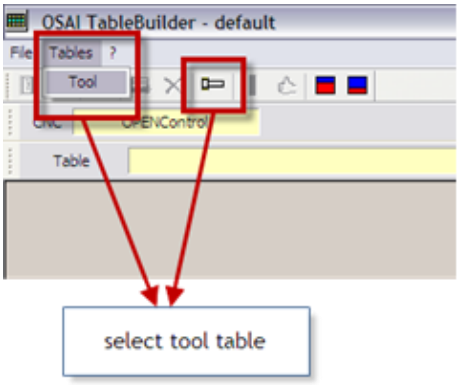


9.3 Tool Table customization

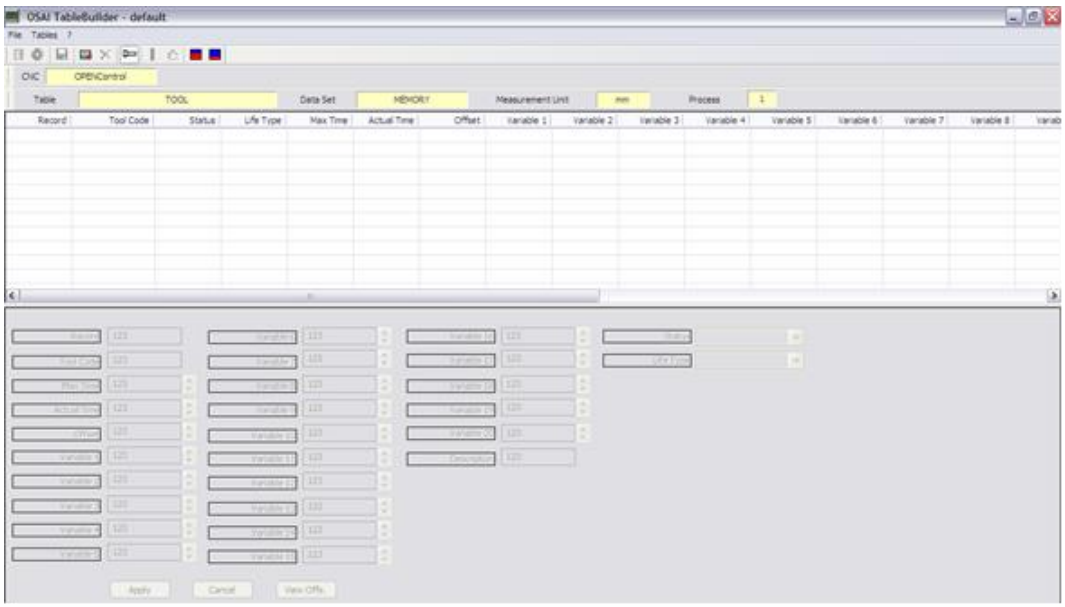
In the upper side of the TABLES DEFINITION window a check box enables/disables the TOOL TABLE view.



Following confirmation with the OK button from the Tool Bar or from the dedicated button, select the tool table

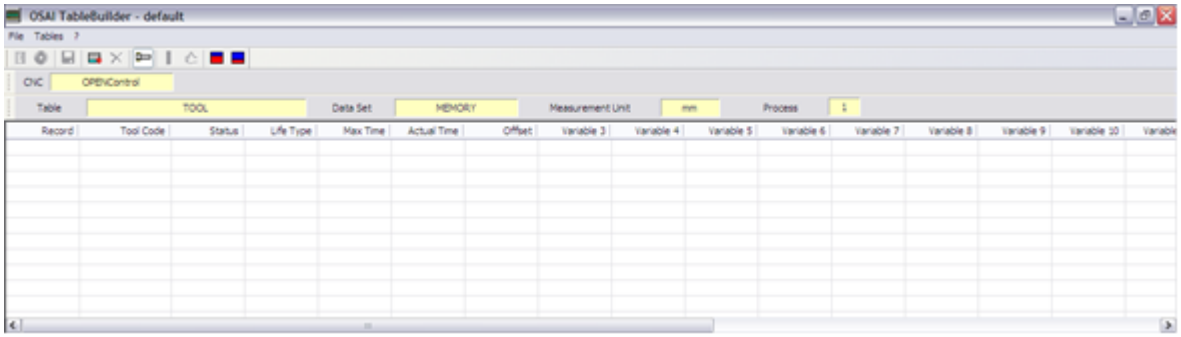


that is displayed as the following picture.

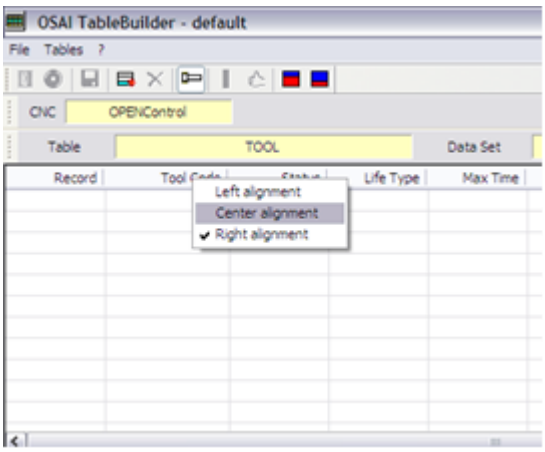


To change the fields displayed in the table above, right-click on the table. A window opens where, by using a check box, it is possible to enable/disable the display of the different columns. For instance, in the following image, the VARIABLE1 and VARIABLE2 display is disabled.

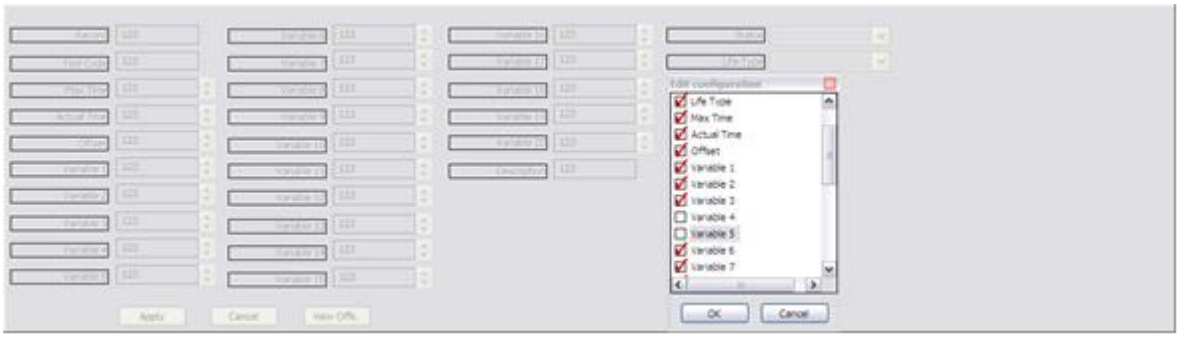
After confirming with the "OK" button, the table is displayed as in the image below. The columns VARIABLE 1 and VARIABLE 2 are no longer visible.



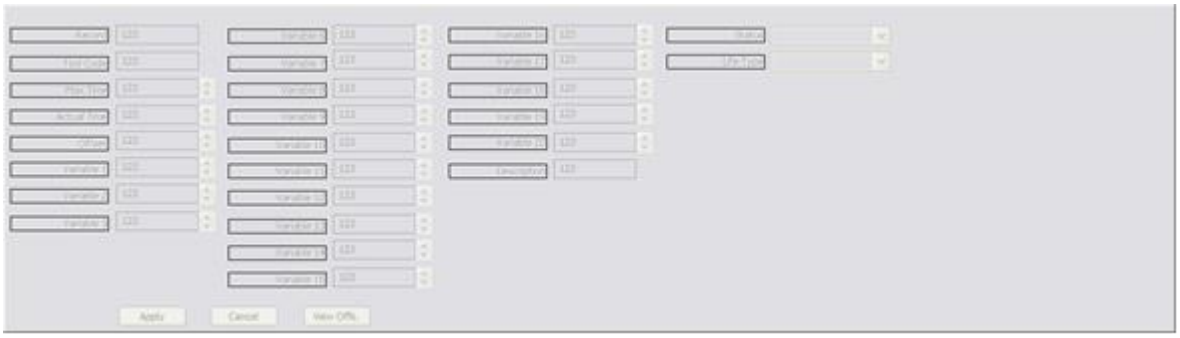
It is also possible to change the alignment of the fields of each column: right-click on the column heading and set the preferred choice. In the image below, for instance, the alignment of the second column (TOOL CODE) is changed from RIGHT to CENTRE.



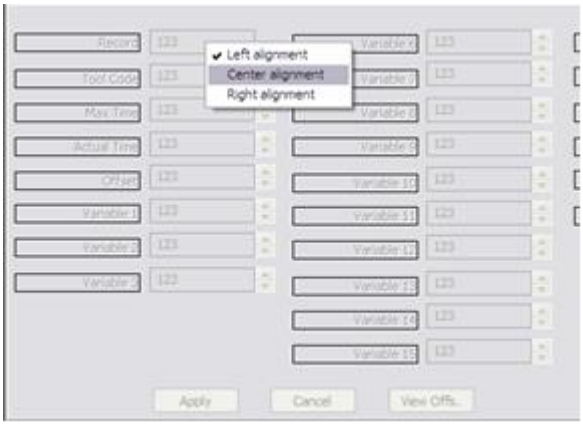
By right-clicking any point of the low window, it is possible to define which fields may be viewed and changed by the end user. In the image below, for instance, the variables VARIABLE4 and VARIABLE5 are made invisible by removing the related check.



Following the “OK” confirmation, the two variables disappear from the screen.

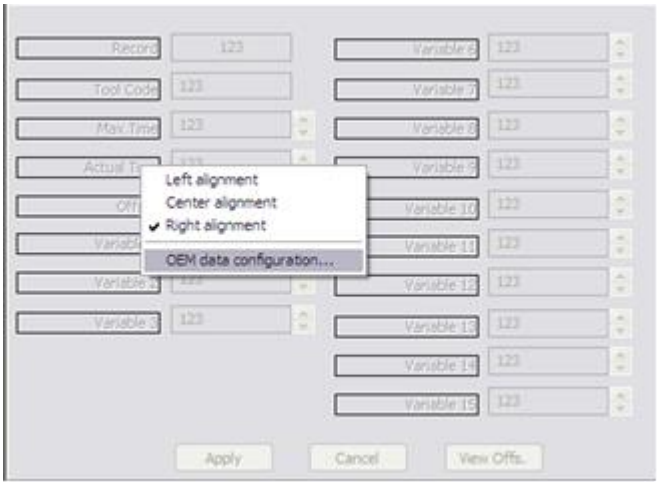


It is also possible to define the alignment of fields in the lower data insert window. This can be done with a right-click on the selected field. For instance, in the image below, the user wants to change the alignment of the field RECORD from LEFT to CENTRE.

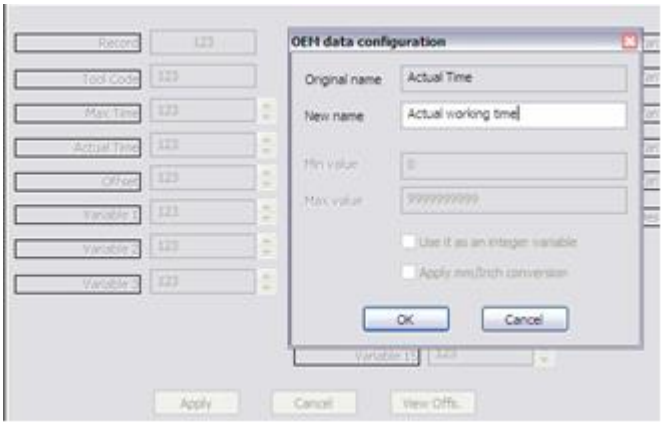


Following the change, the field appears as in the image below.

Another possibility is to define some features for every single field. This can be done with a right-click on the field label and then selecting OEM DATA CONFIGURATION.

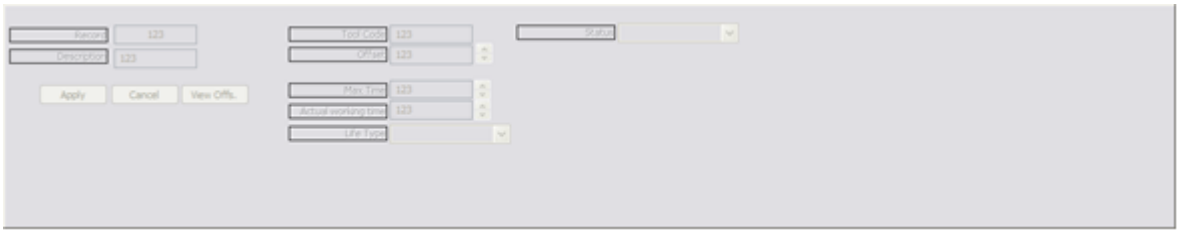


The OEM DATA CONFIGURATION window is opened and it is possible to change the variable name (in the case of tools, only the name can be changed). For instance, in the image below, the user wants to change the label name referred to the ACTUAL TIME variable.

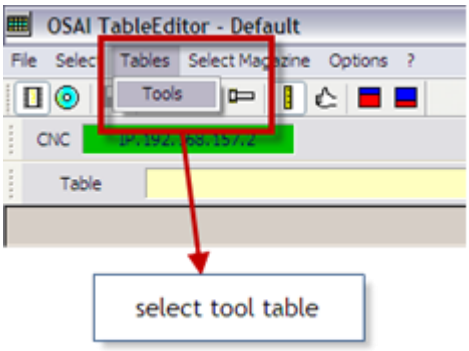


After the "OK" confirmation, the variable will be displayed as in the image below.

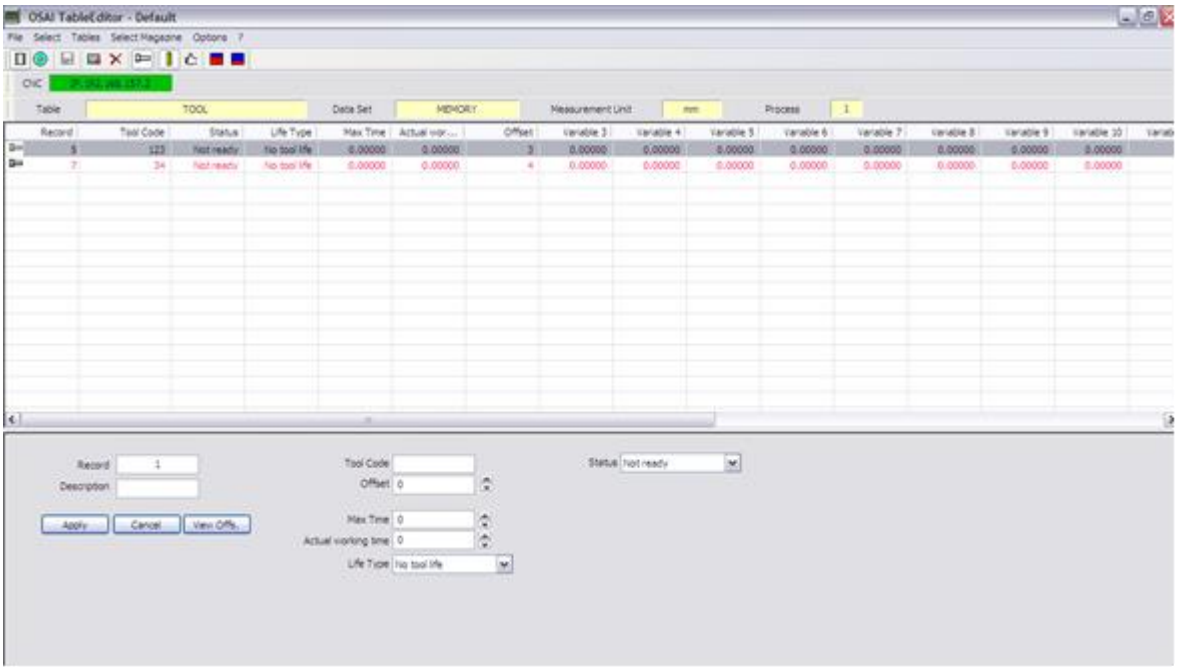
The fields in the window can also be placed in a different position by using drag & drop. When you select a field connected to others (for instance label, value and spin-button) the three elements are outlined by a small red square. In the image below, the fields are set in a different layout.



The final result of the changes can be viewed from Table Editor by selecting the Tool Table from the Tool Bar or by the appropriate button.

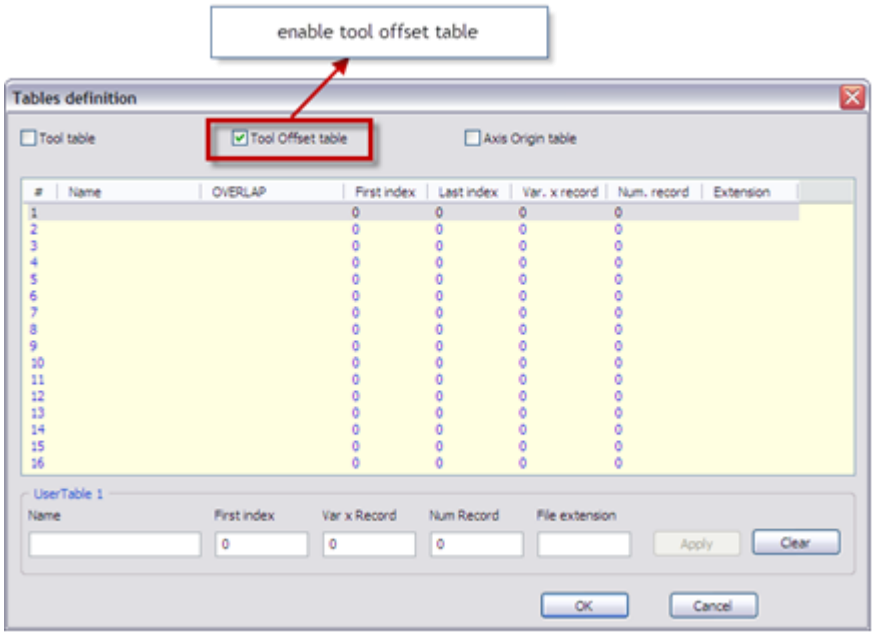


The following window is displayed where only the tool data enabled according to the configuration made in this example is displayed.

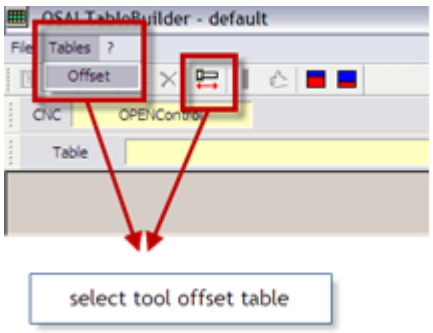


9.4 Table Offset Customization

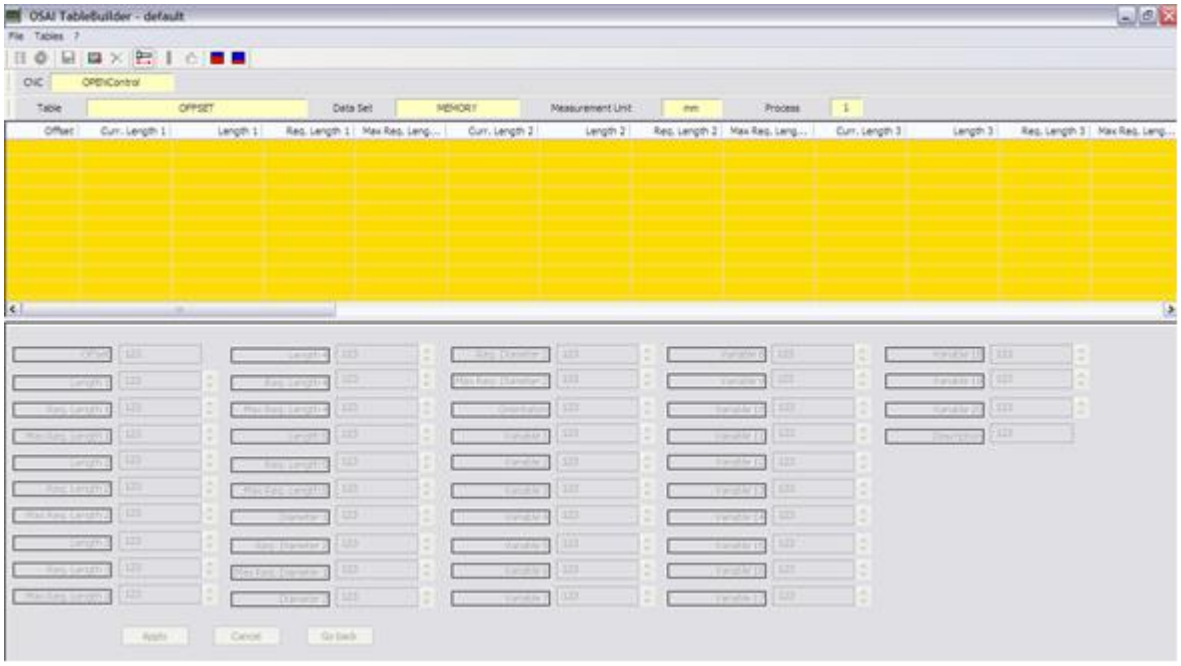
In the upper section of the TABLES DEFINITION window it is possible to enable/disable the TOOL OFFSET TABLE display.



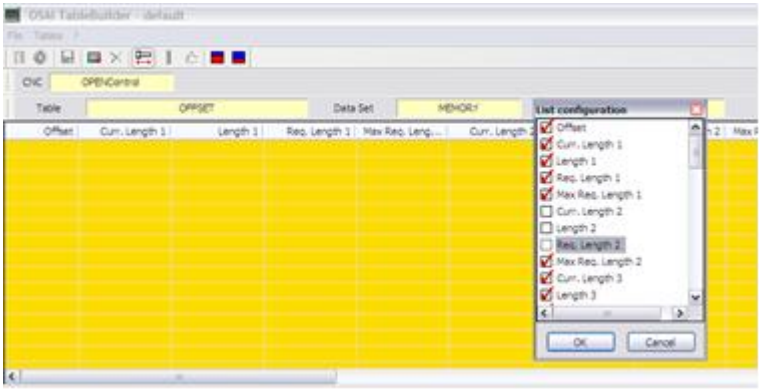
After confirming with “OK” from the Tool Bar or from the appropriate button, it is possible to select the tool offsets table



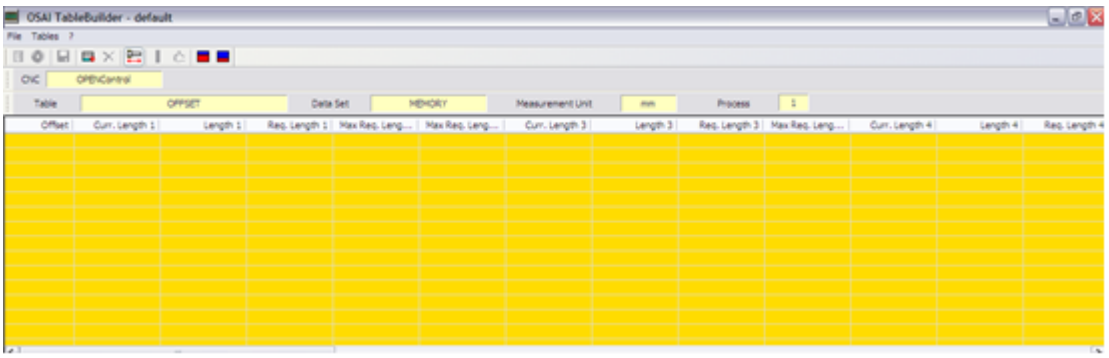
as displayed as in the image below



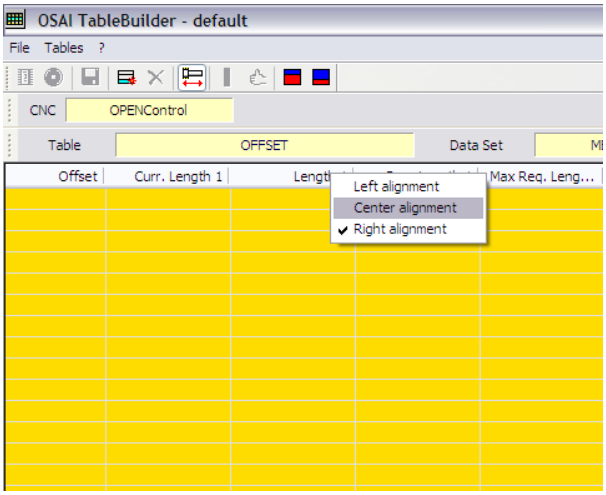
To change the fields display in the table above, right-click on the table. A window appears where, by using a “check”, it is possible to enable/disable the columns display. For instance, in the image below the display of the variables CURR.LENGTH2, LENGTH2 and REQ. LENGTH2 is disabled.



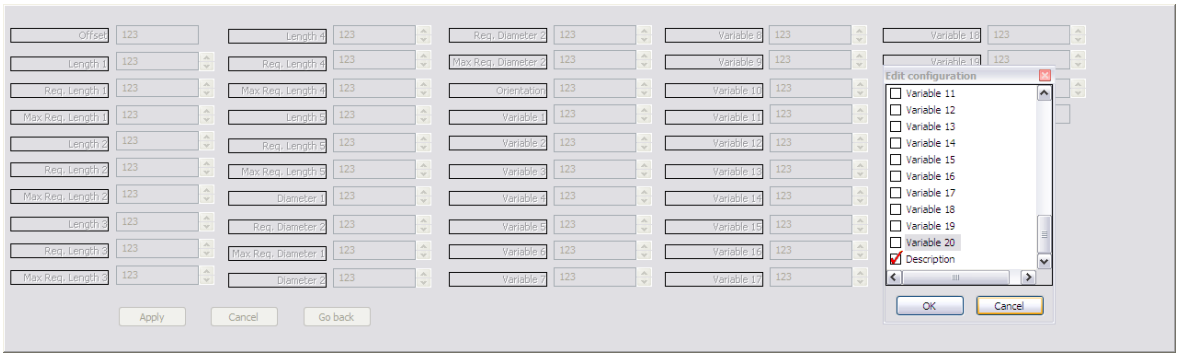
Following the confirmation with the “OK” button, the table is displayed as the image below: the columns CURR.LENGTH2, LENGTH2 and REQ. LENGTH2 cannot be viewed anymore.



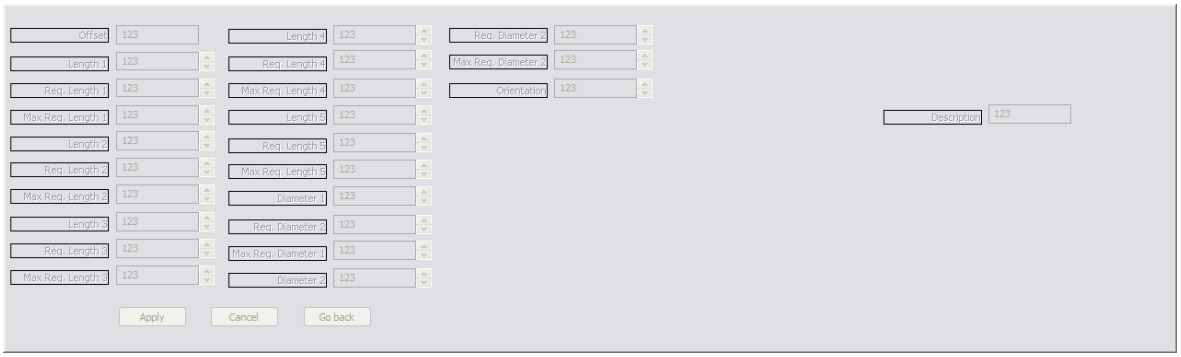
It is also possible to change the alignment of the fields of every column; this can be done by right-clicking on the column heading and then selecting the desired choice. In the image below, for instance, the alignment of the second column (LENGTH1) is changed from RIGHT to CENTRE.



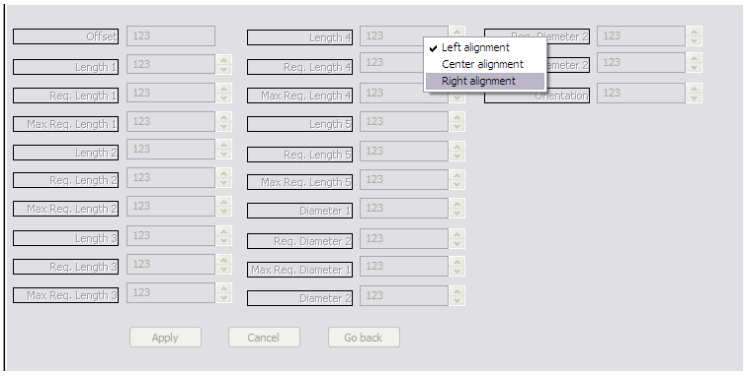
By right-clicking any point in the lower data insert window, the fields may be viewed and changed by the user may be defined. In the image below, for instance, the variables from VARIABLE1 to VARIABLE20 are made invisible simply by removing the associated check.



Following confirmation with the “OK” button, the variables are no longer displayed.

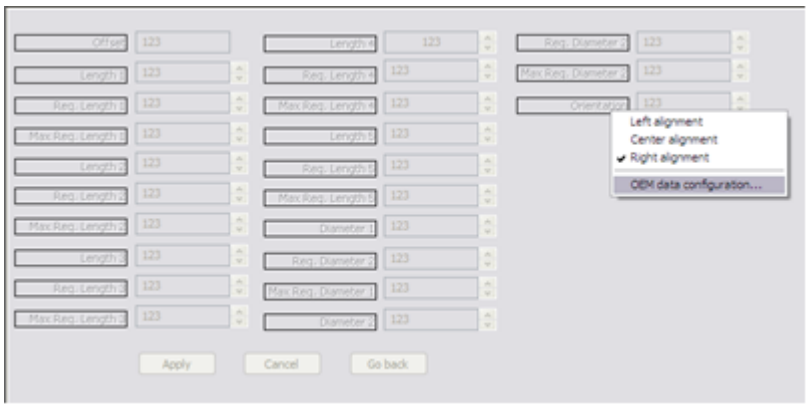


It is also possible to define the alignment of each field in the lower insert data window. Right-click on the field and then select using the "check". In the image below, for instance, the user wants to change the field LENGTH4 alignment from LEFT to RIGHT.

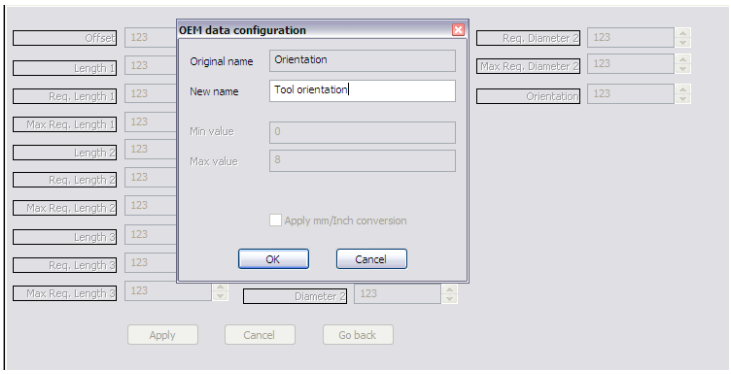


After the change, the field appears as shown below.

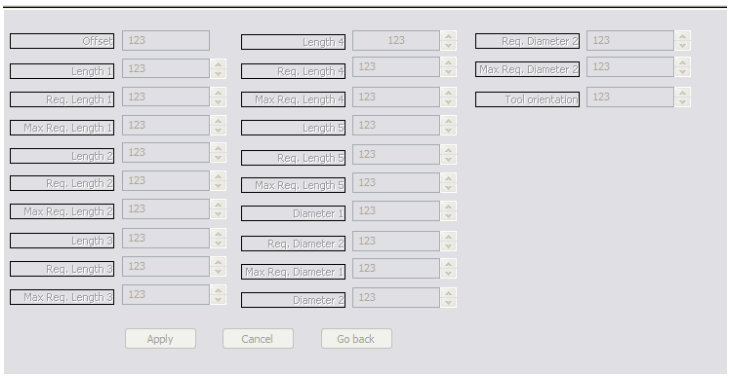
Another possibility is to define some features for each field. This can be done by right-clicking on the field label and then selecting MTB DATA CONFIGURATION.



The OEM DATA CONFIGURATION window opens: here the name of the variable can be changed (in the case of tool offsets only the name can be changed). In the image below, for instance, the user wants to change the name of the label associated with the variable ORIENTATION.

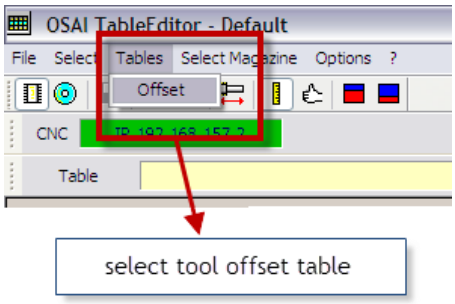


After confirming with the “OK” button, the variable will be displayed on the screen as shown below.

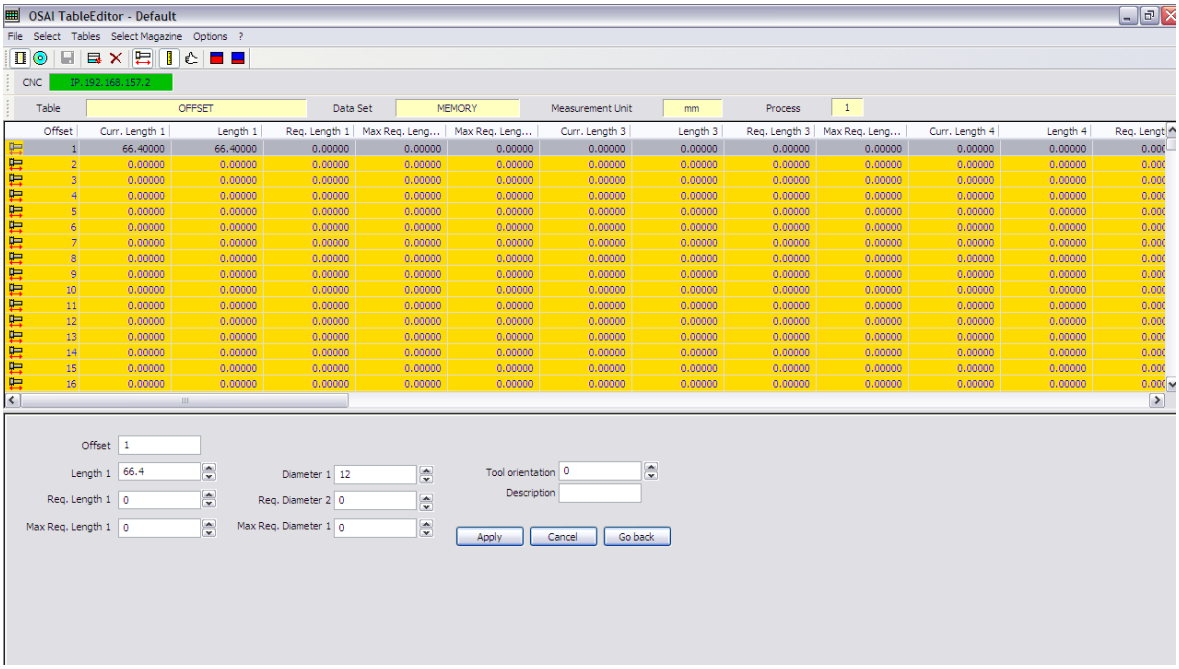


The fields in the window can also be positioned in a different way by using drag & drop. When you select a field connected to others (for instance label, value and spin-button) the three elements are outlined by a small red square. In the image below, the fields are arranged in a different way.

The final result of the changes can be viewed when, from Table Editor, the Tool Offset Table is selected from the Tool Bar or by the appropriate button



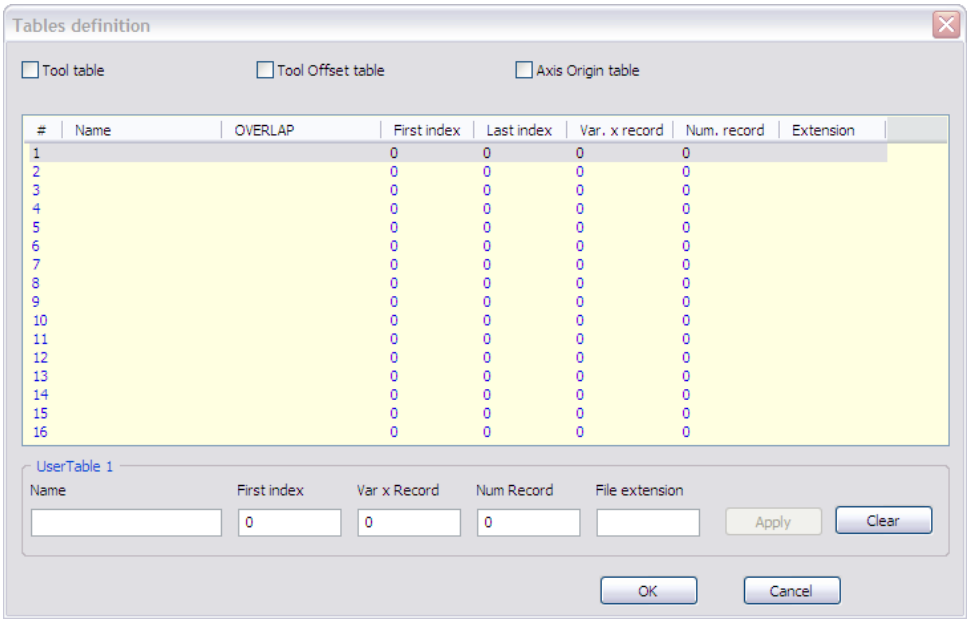
the window below is displayed, here only the data enabled by the configuration for display is shown.


OSAI TableEditor - Default
File Select Tables Select Magazine Options ?
CNC IP.192.168.157.2
Table OFFSET Data Set MEMORY Measurement Unit mm Process 1
Offset Curr. Length 1 Length 1 Req. Length 1 Max Req. Leng... Max Req. Leng... Curr. Length 3 Length 3 Req. Length 3 Max Req. Leng... Curr. Length 4 Length 4 Req. Lengt
Offset 1
Length 1 66.4
Req. Length 1 0
Max Req. Length 1 0
Diameter 1 12
Req. Diameter 2 0
Max Req. Diameter 1 0
Tool orientation 0
Description
Apply
Cancel
Go back

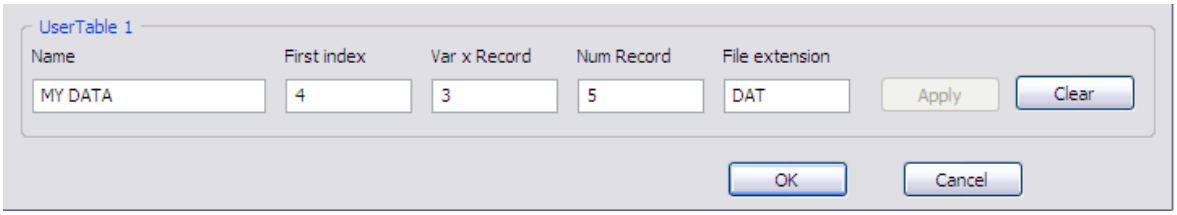
9.5 User Table customization

In the lower part of the TABLES DEFINITION window, the grouping of the 400 variables of the USER table can be configured in up to 16 different tables in which the following items can be configured:

- table name
- first variable index
- number of variables per record
- record numbers
- the file extension with which this table will be saved

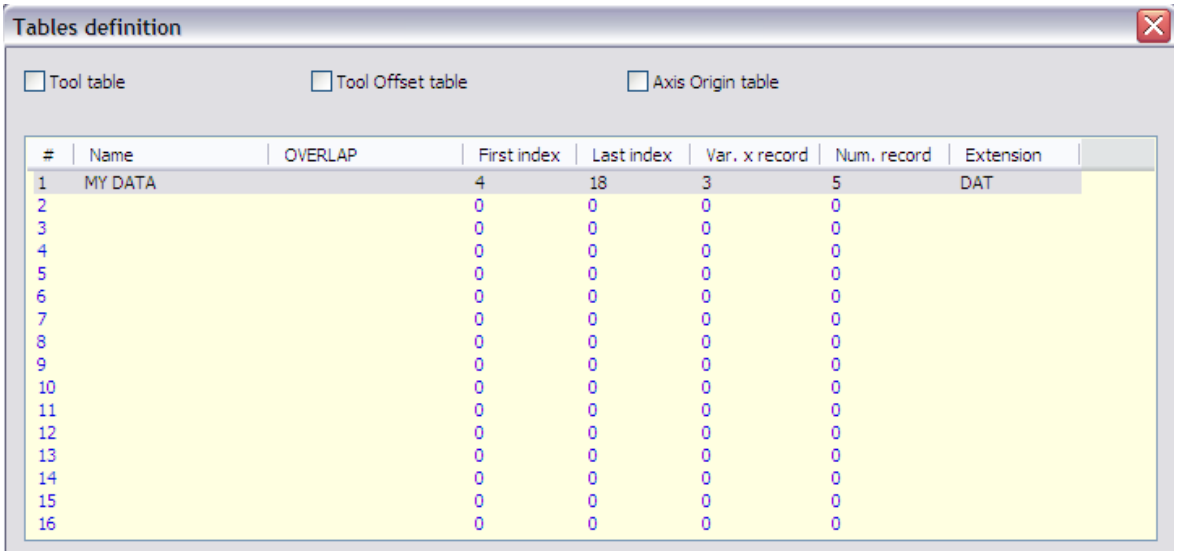


EXAMPLE: to define table 1 with the name “MY DATA” starting from variable 4, having 5 records with 3 variables per record: this data must be saved in a file with the extension DAT.



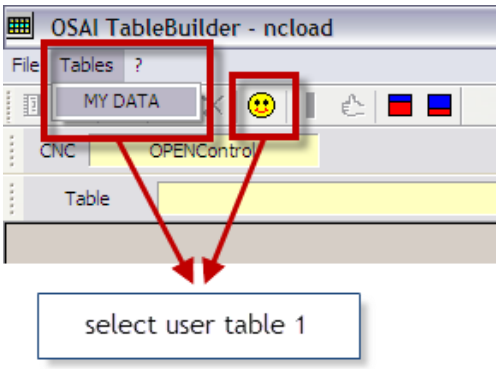
The fields must be filled in as described in the following figure.

By pressing the APPLY button, the data will be saved in table 1.

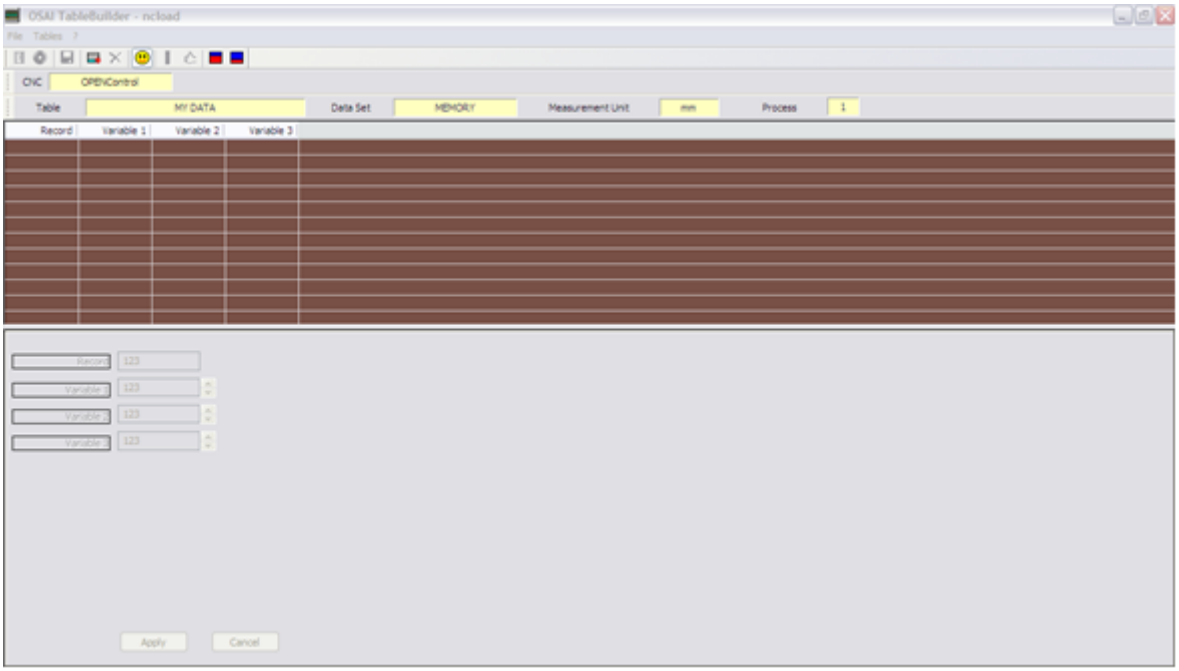

Tables definition
Tool table
Tool Offset table
Axis Origin table
Name OVERLAP First index Last index Var. x record Num. record Extension
1 MY DATA 4 18 3 5 DAT

To delete a table previously defined select the corresponding line, press the CANCEL button and confirm deletion with the next request message.

At this point, in the Tool Bar, first select TABLES and then the name of the user table - MY DATA -just inserted or by pressing the dedicated button



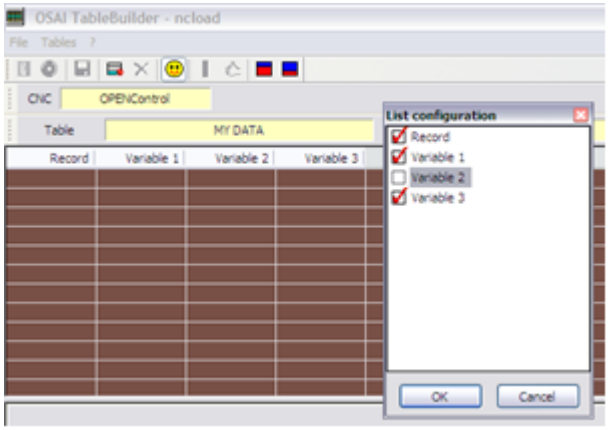
and the window below appears.



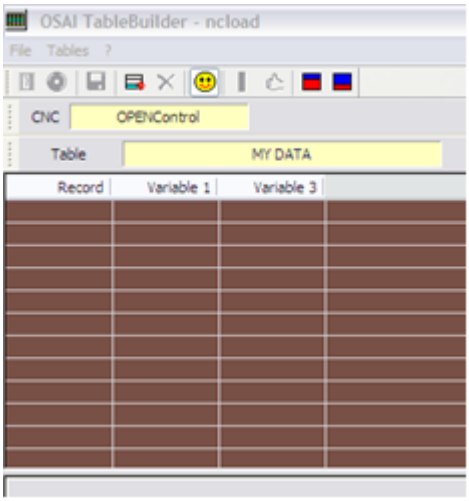
In this ambient, the following fields can be defined:

- the fields to be displayed
- the fields that can be changed by the end user
- the name, the format, the minimum/maximum value, the possible dependence of each parameter on the selected measurement
- the position of the record in the lower data insert window

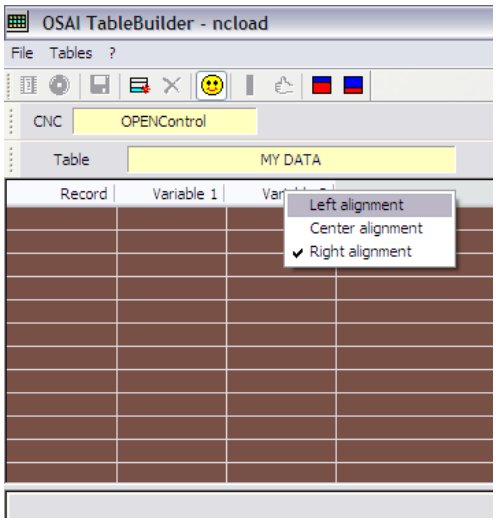
Right-click on the table to change the fields display. A window opens in which, by means of a “check” box, display of the entire column can be enabled / disabled. For instance, in the following window disable the display of the column indicated with VARIABLE 2.



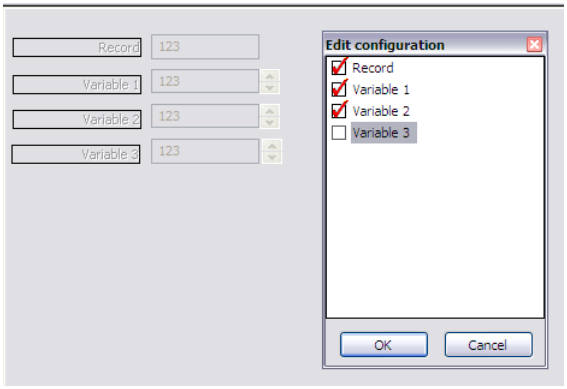
After confirming with the “OK” button, the table is displayed as shown in the following figure: VARIABLE 2 column is no longer visible.



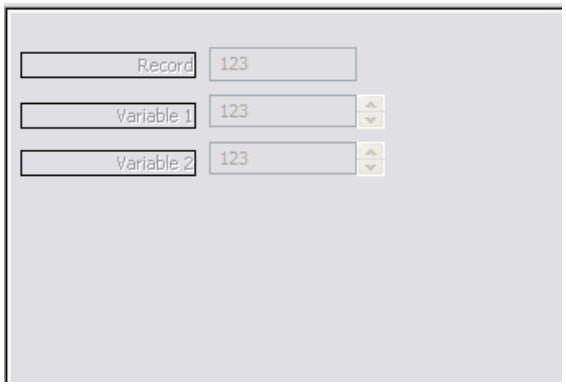
It is also possible to change the alignment of fields for each column: right-click on the column heading and then select the desired choice with the “check”. In the figure below, for instance, the alignment of the third column (VARIABLE 3) is changed from RIGHT to LEFT.



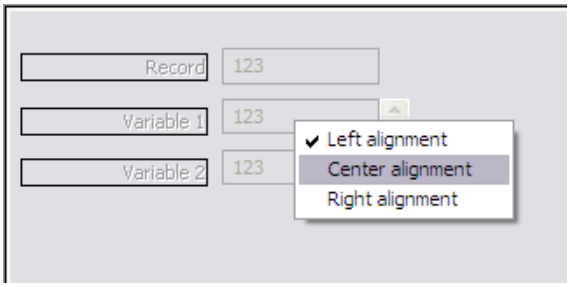
Right clicking anywhere in the lower data entry window allows which fields can be viewed and modified by the end user to be selected. In the following figure, for example VARIABLE3 is hidden by removing the "check".



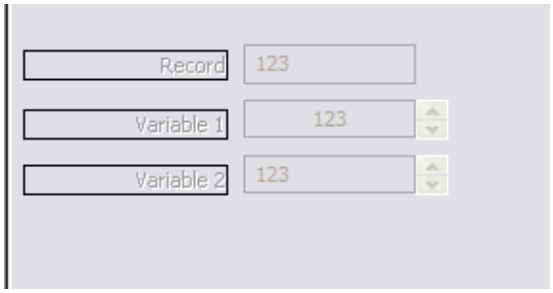
After confirming with the "OK" button, the third variable disappears from the screen.



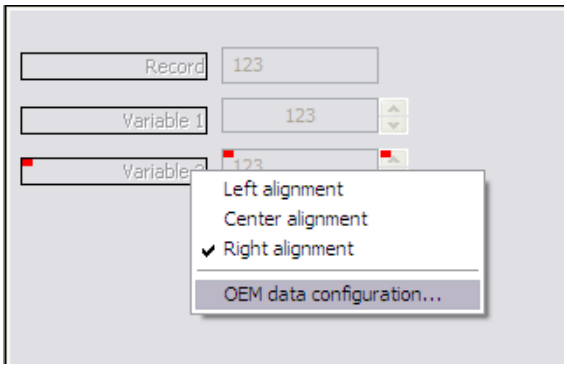
The alignment of each field in the lower data entry window can be defined. Right-click on the field to select. For instance, in the following figure change the alignment of the VARIABLE1 field from LEFT to CENTER.



After the change, the field appears shown below.

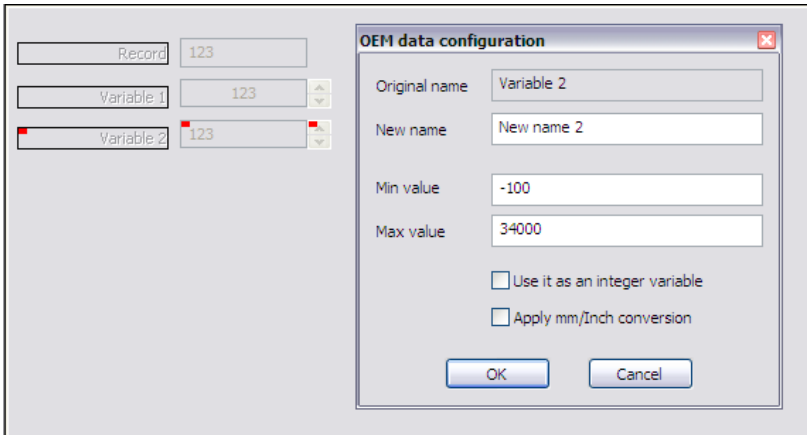


Another possibility is to define some features of every field. This can be done by clicking the label of the field and selecting MTB DATA CONFIGURATION.

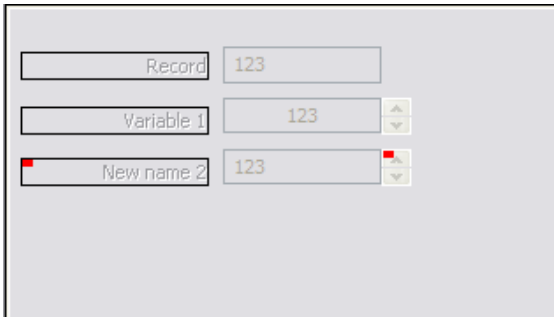


The window MTB DATA CONFIGURATION opens and here it is possible:

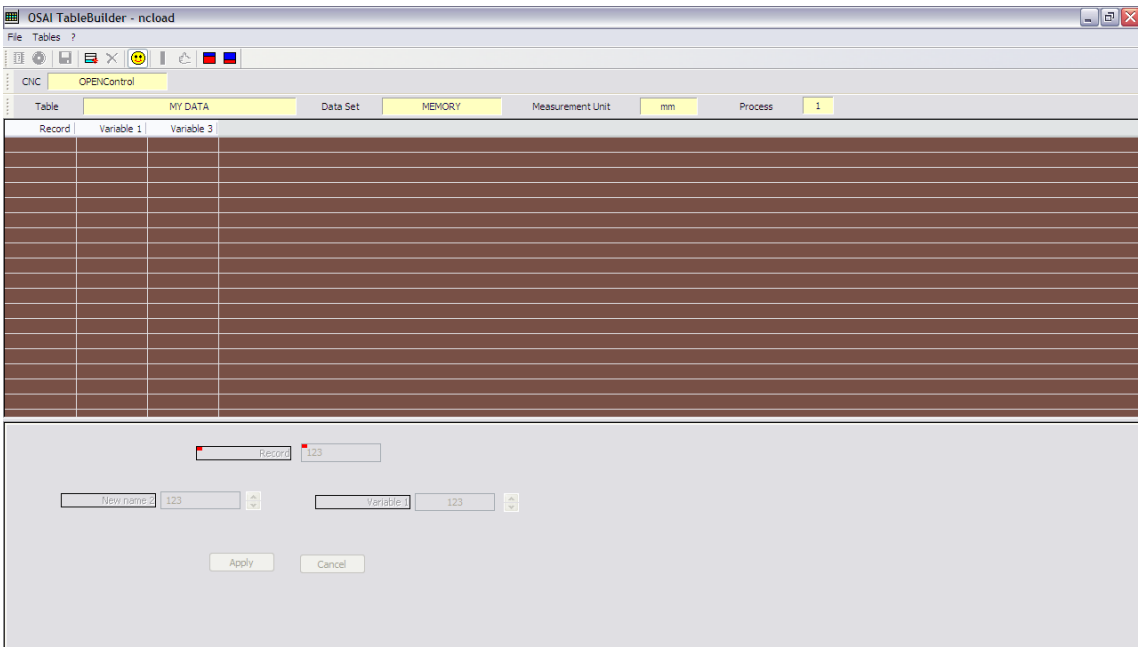
- to change the variable names
- to set the maximum/minimum variable value
- to define the variable as INTEGER
- to enable automatic conversion when the measurement unit changes



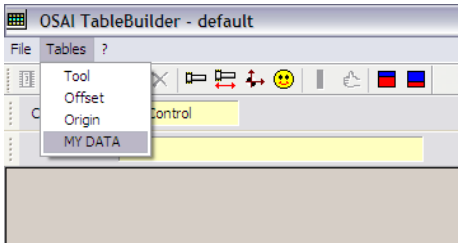
After confirming with the OK button, the variable will appear on the screen as shown below.



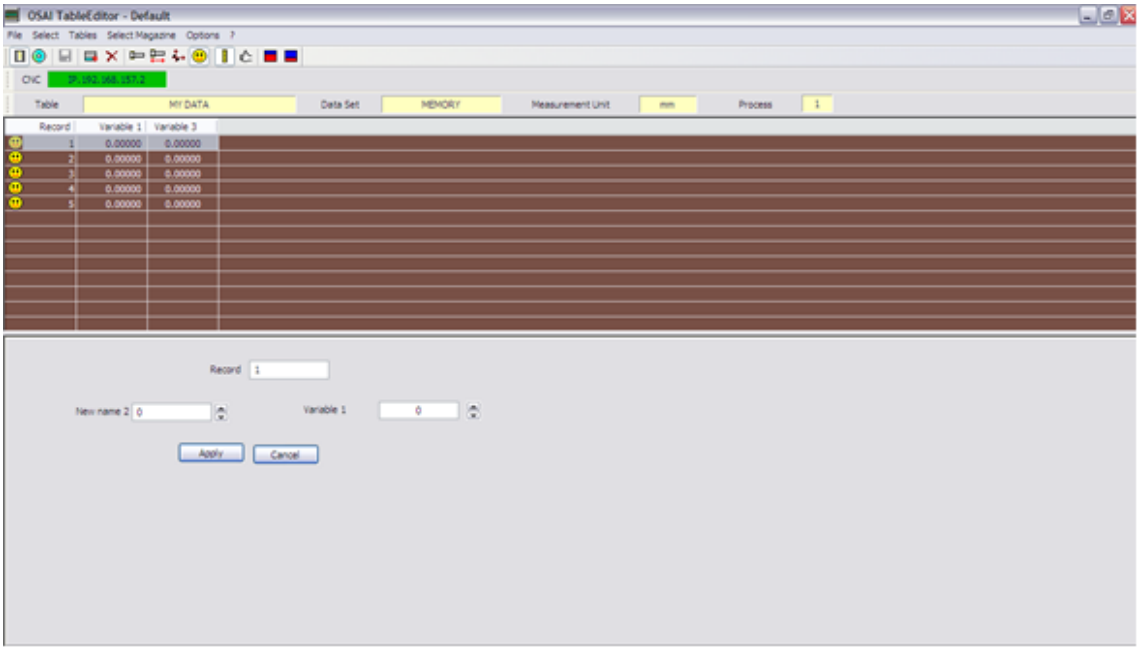
The fields in the lower window can be positioned in a different way by using a drag & drop. When if a field is selected that is related to another (e.g. label, value and spin-button) the three related elements are highlighted by a small red square. In the following figure the fields were positioned in a different graphical layout.



The final result of the changes made, is visible when the MY DATA table is selected from the Table Editor, or the appropriate button is pressed



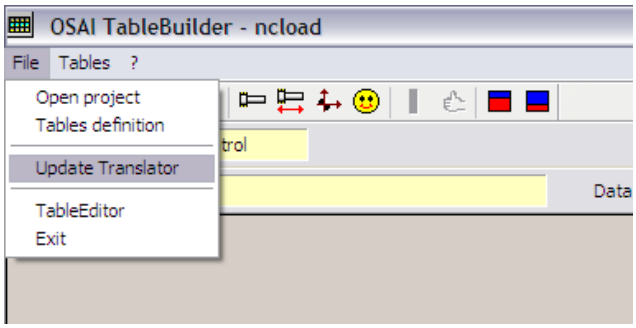
when the following window appears.



If more than one user table is configured, the same variables can be configured in different tables. This is allowed and it is not displayed as an error.

9.6 Translation of the variable names

The names of the fields that belong to the standard tables are automatically translated into the language selected by NBI Config. But this cannot be done for those variables whose names have been changed: to allow the translation of these names the UPDATE TRANSLATOR command present in the Tool Bar is used.



When this command is launched, it is automatically creates/ updates the corresponding CSV file in the first column contains the list of names of all the changed fields. If a new field is added, it is reported in the CSV file when the command is executed. If a field is deleted and is no longer used, it is still listed at the end of the CSV file but is marked as UNUSED.

This CSV file can be changed using a text editor or using Microsoft Excel, and looks as follows:

```
****** Do not open this file by double clicking its file name. ******
****** Please Use the EXCEL "File - Open" menu command.     ******
****** Do not delete these three text lines.                ******
Keyword;       English; French
Counter:
X axis limit;
Level;
Program;;
```

After the first 3 lines of text (which must not be modified or deleted), the fourth line begins with the key KEYWORD indicating the start of the (first) column that contains the field names added in the original language that may have to be translated into other languages: in the next line, always in the first column, automatically reported by the UPDATE TRANSLATOR command are listed all the names of the fields that are created / edited / deleted. In the fourth line, to the right of KEYWORD and separated by the character ";", other keys can be added (in the example ENGLISH and FRENCH) identifying the language of the words written in the following lines in the second and third columns respectively.

Starting from the fifth row, and then for each subsequent row, will be added by the operator, in the second and third columns, the translation in the languages listed, separated by the character ";"
If a language other than those provided (ENGLISH and FRENCH in our example) is selected, the field in the first column will be displayed.
If the field translation is missing, the field in the first column is displayed.

9.7 Configuration file

All files associated with the layout and configuration of the tables created by the Tables Builder are saved in the following folder of the PC hard disk

C:\Documents and Settings\All Users\Application Data\OSAI\WinNBI\Config\TableEditor**PROJECT_NAME**

where **PROJECT_NAME** is the name given to the project.

10. Tool Management

10.1 General

Tool Management is a series of procedures allowing the parameters to be defined and changed as described in the previous chapters concerning the tools. The standard functions available for tool parameter management are:

- display and modify tool offsets
- cancel tool offsets
- apply incremental modifications to tool offsets
- add new tools to the Tools table
- display and modify tools
- cancel tools.

Another important function for the tool management is PRESETTING, described in this chapter.

The functionality described in this chapter, follows the standard operating modes. Different operating modes can be set by the MTB, these require specific instructions to be defined by the machine tool manufacturer.

10.2 Tool Pre-setting

The PRESETTING tool measures the tool length and stores it in the Tool Offset Table.

Before starting the pre-setting procedure, make sure to reference the spindle axis (typically Z).

Tool pre-setting is not available when the TCP functionality is active.

Tool PRESETTING can be made through the procedures described below:

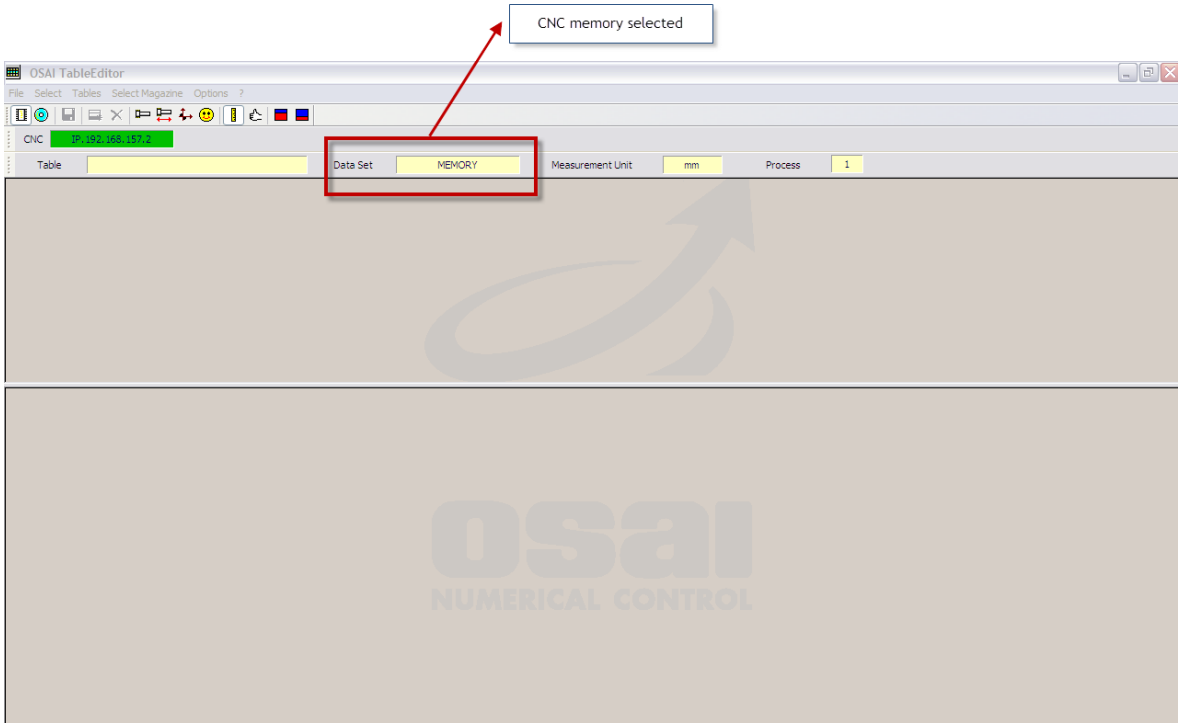
1) Manual Tool pre-setting
2) Manual pre-setting of the tool mounted in the spindle
3) Automatic pre-setting of the tool mounted in the spindle

10.2.1 Manual Tool pre-setting

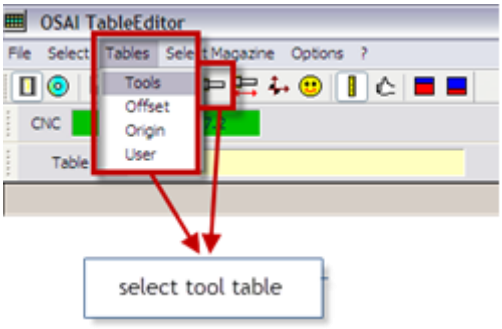
Using this procedure, the tool dimensions (assuming they have been obtained by measurement with an external system) can be written directly into the CNC working area. The same procedure can be used also to insert, change or delete any tool parameter.

Launch the Table Editor by selecting “UTILITY” in the Tool Bar and then TABLE EDITOR or use the dedicated button.

The window below appears. To change the CNC working area, first check if “MEMORY” is selected.



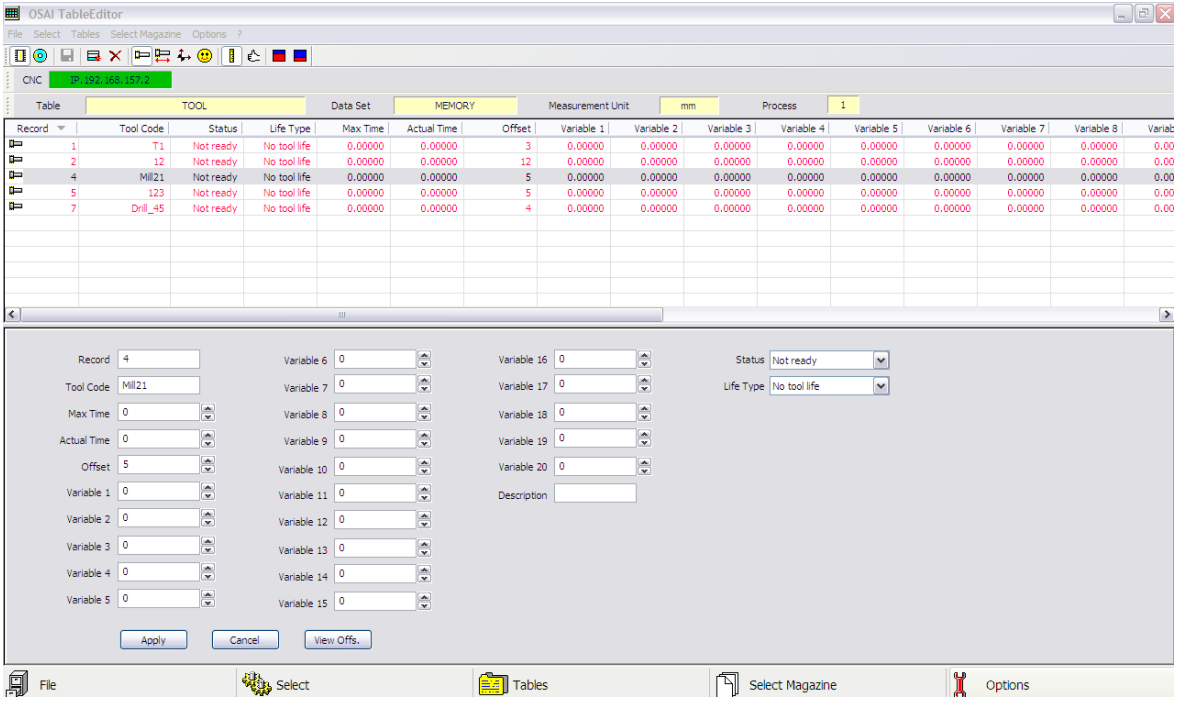
Launch “TABLES” from the Tool Bar and then TOOLS, or use the dedicated button.



At this point you will see the following dialog window which lists all the tools configured in the CNC memory area. Select the desired tool whose code can be read in the TOOL CODE column.

As an example, the following figure shows 5 tools configured in the memory area of the CNC. The tool

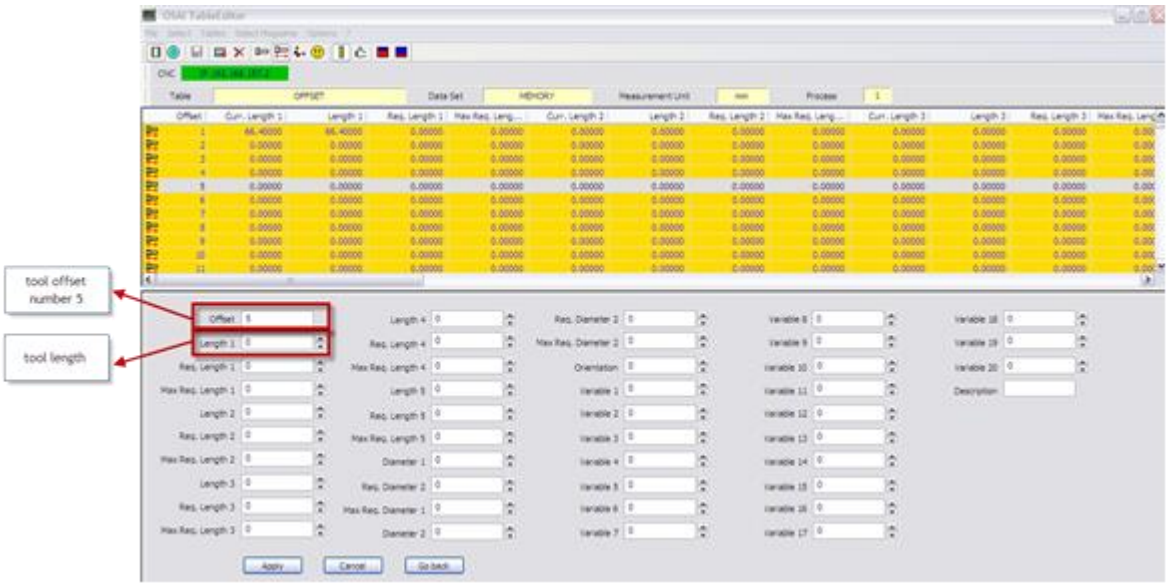
selected has the MILL21 code which is associated to offset 5. All the parameters of the selected tool are displayed at the bottom of the screen.


OSAI TableEditor
File Select Tables Select Magazine Options ?
CNC IP.192.168.157.2
Table TOOL Data Set MEMORY Measurement Unit mm Process 1
Record Tool Code Status Life Type Max Time Actual Time Offset Variable 1 Variable 2 Variable 3 Variable 4 Variable 5 Variable 6 Variable 7 Variable 8
1 T1 Not ready No tool life 0.00000 0.00000 3 0.00000 0.00000 0.00000 0.00000 0.00000 0.00000 0.00000 0.00000
2 12 Not ready No tool life 0.00000 0.00000 12 0.00000 0.00000 0.00000 0.00000 0.00000 0.00000 0.00000 0.00000
4 Mill21 Not ready No tool life 0.00000 0.00000 5 0.00000 0.00000 0.00000 0.00000 0.00000 0.00000 0.00000 0.00000
5 123 Not ready No tool life 0.00000 0.00000 5 0.00000 0.00000 0.00000 0.00000 0.00000 0.00000 0.00000 0.00000
7 Drill_45 Not ready No tool life 0.00000 0.00000 4 0.00000 0.00000 0.00000 0.00000 0.00000 0.00000 0.00000 0.00000
Record 4
Tool Code Mill21
Max Time 0
Actual Time 0
Offset 5
Variable 1 0
Variable 2 0
Variable 3 0
Variable 4 0
Variable 5 0
Variable 6 0
Variable 7 0
Variable 8 0
Variable 9 0
Variable 10 0
Variable 11 0
Variable 12 0
Variable 13 0
Variable 14 0
Variable 15 0
Variable 16 0
Variable 17 0
Variable 18 0
Variable 19 0
Variable 20 0
Description
Status Not ready
Life Type No tool life
Apply Cancel View Offs.
File Select Tables Select Magazine Options

The tool code can be separated from the offset number.

Please note that the tool parameters related to the tool state and to tool life, are only meaningful when managed by the machine logic developed by the OEM otherwise they are ignored.

For example, to change the length of the selected tool, select the display of the parameters of the associated offset. For this, simply press the VIEW OFFS button to display the Offset Editor with the offset 5 already selected.

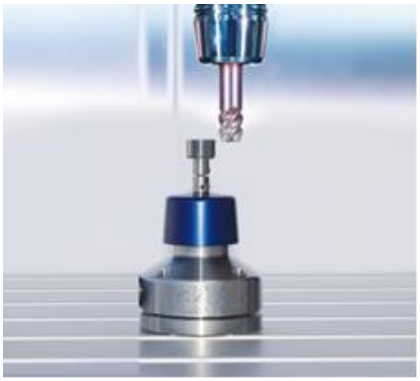


All the parameters of the offset are displayed in the lower window and can be changed. In this example, the length LENGTH1 is modified by entering the desired value. After pressing the APPLY button, the change is saved in the CNC working area and is immediately effective.
At this point return to the tool table by pressing the GO BACK button, or close the offset table and exit.

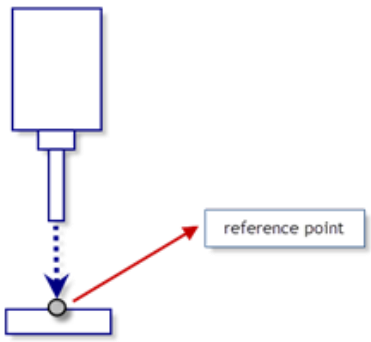
Note that different tools can have the same offset. Modifying such an offset means the change will be applied to all tools that use it.

10.2.2 Manual pre-setting of the tool length mounted on the spindle

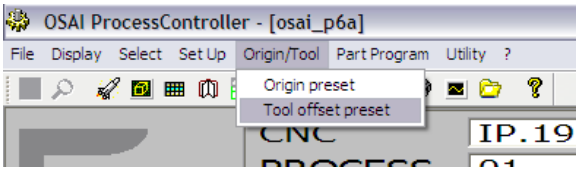
The length value of a tool mounted on the spindle can be measured on the machine and written in the table automatically, i.e. without opening the table editor. The tool is measured with respect to a predetermined reference point.



Assume that a theoretical length value has already been set for the tool in the spindle, which is associated with, for example, the offset 1,and this value is to be checked. Activate offset 1 and physically position the tool tip, along the Z axis, on a reference point whose dimension is known (for convenience this is generally Z0).



In this condition, the value displayed on the screen for the Z axis should be Z0. However if a value different from what was expected (e.g. Z+0.4): this means that the pre-set value of tool length is not correct. This value can be corrected by selecting ORIGIN/TOOL on the Tool Bar and then TOOL OFFSET PRESET.



The TOOL OFFSET PRESET window appears where the following data must be set:

- OFFSET NUMBER: the offset number to be changed (1 in the example)
- REFERENCE AXIS: the axis to which the tool is directed (Z in the example)
- TOOL TIP POSITION: the Z position of the tool tip (Z0 in the example)

If “0” is set for the TOOL TIP POSITION, after confirming with the “OK” button, the tool length (offset 1) is updated from the previous value (offs1 = 66 in the example) to the new correct value (66.4), this can be verified by opening the offset table editor. Having thus updated the value, and as it is currently active, the Z coordinate on the screen is immediately updated to the new value (Z0) which takes into account the correct length of the tool.

10.2.3 Automatic pre-setting of the tool mounted in the spindle

Alternative cycles of automatic tool setting, regarding both the length and the diameter, can be prepared by the OEM and can be used after specific instructions from the machine manufacturer.

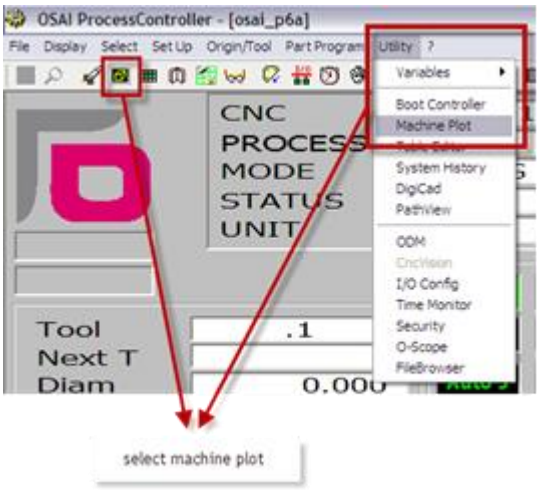
11. Machine Plot

11.1 General

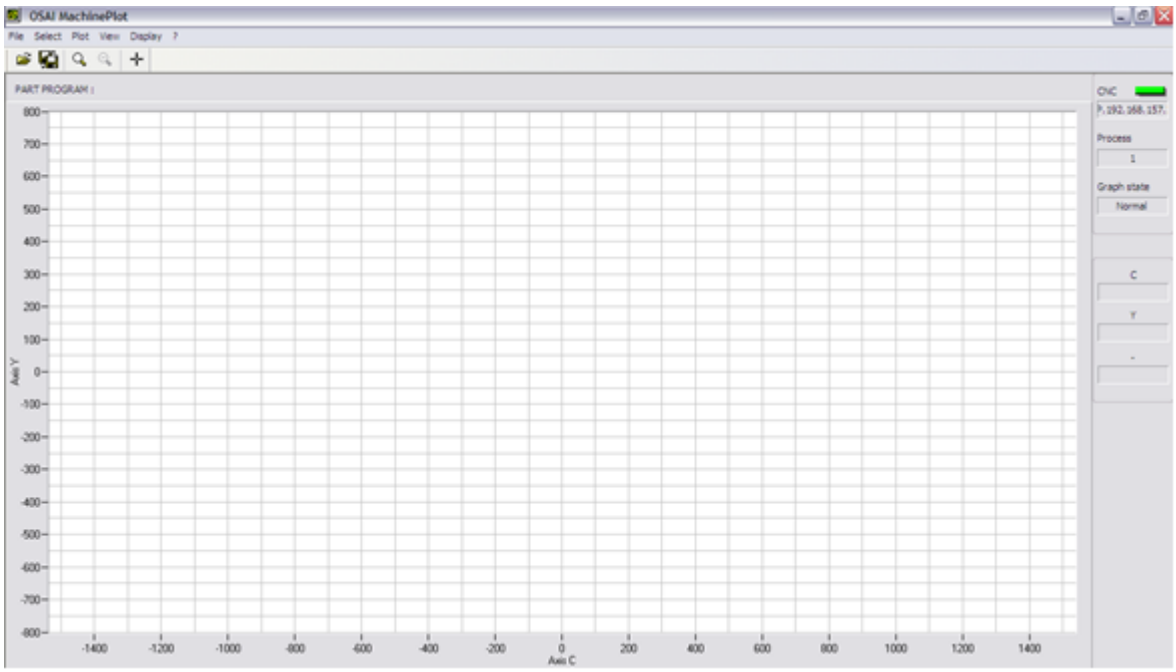
The Machine Plot is a tool for debugging the user part program that displays the tool path both during actual machining and simulation (DRY RUN). In the first case, in the 2D display, both the theoretical path and the actual path are displayed, in the case of a simulation only the theoretical path can be seen.

11.2 Machine Plot Interface

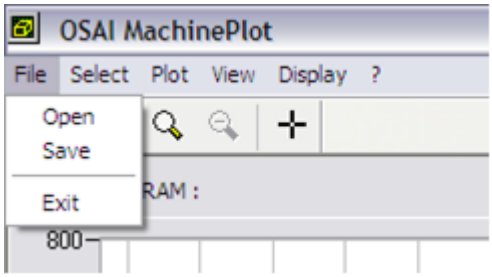
The Machine Plot can be enabled from the Tool Bar, by selecting "UTILITY" and then "MACHINE PLOT", or by using the button indicated.



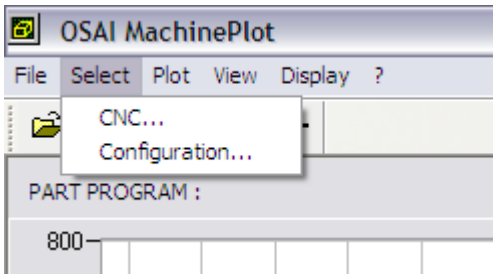
When Machine Plot is launched, the following window appears

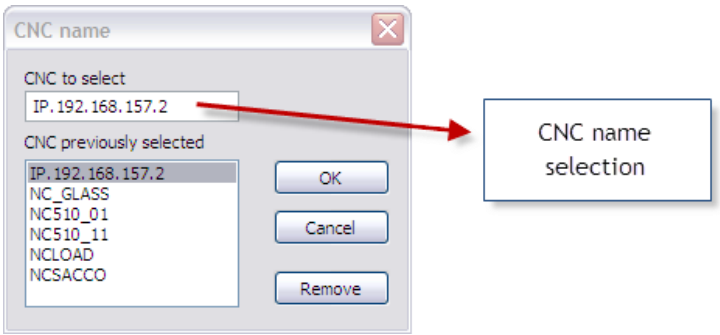


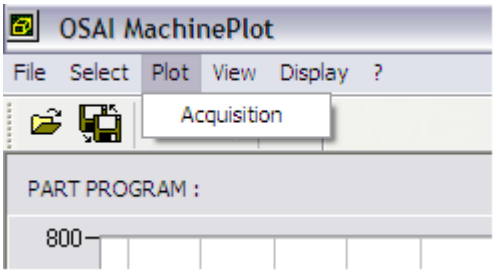
In the Tool Bar table the following commands are available:



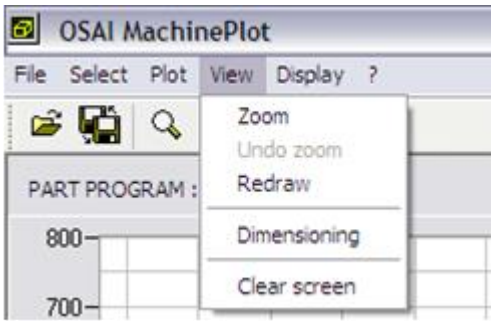
FILE	Open	Opens a file where a profile was stored using the SAVE command
	Save	Save the current profile file. Usually this file is saved on the PC hard disk in the folder C:\Documents and Settings\All Users\Application Data\OSAI\WinNBI A different folder may also be selected. The file name may be selected, but the extension is always MPF.
	Exit	Exit from Machine Plot



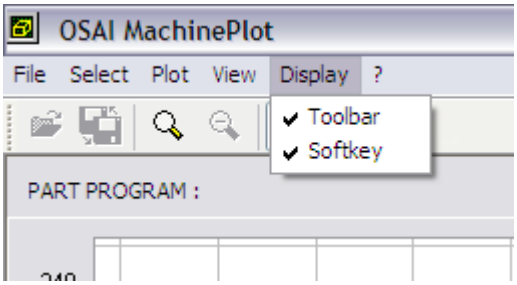
SELECT	CNC	Using this command, the name of the CNC with which the machine plot has to communicate can be defined. This setting is stored when the Machine Plot is closed. The connection with the selected CNC is automatically restored at the subsequent reboot. 
	Configuration	This command opens the CONFIGURATION window where the parameters defining the graphic display profile are set



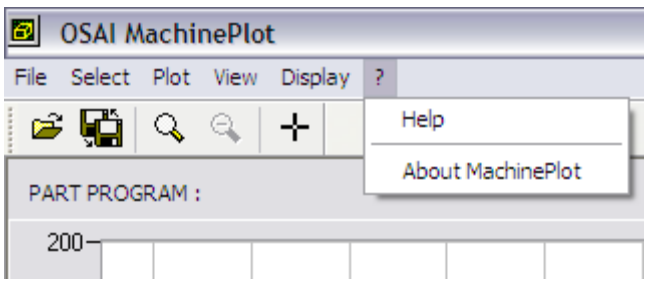
PLOT	Acquisition	This command starts the acquisition of the axis positions and the display of the related tool path.



VIEW	Zoom	This command enables the ZOOM mode that helps in analysing the acquired profile.
	Undo Zoom	This command resets the previous zoomed displays step-by-step
	Redraw	This command restores the video to the original size configured using the CONFIGURATION command and then redraws the profile
	Dimensioning	This command enables the DIMENSIONING mode
	Clear Screen	This command clears the screen



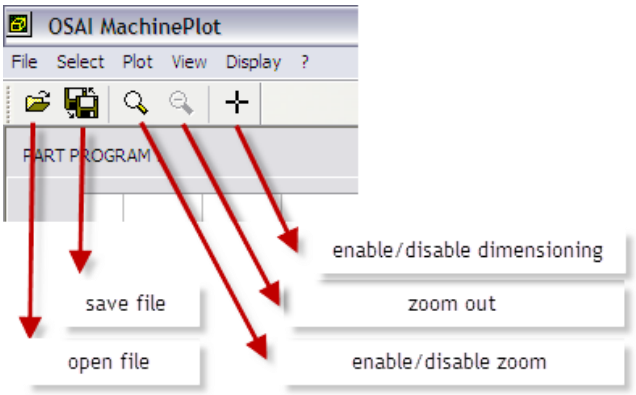
DISPLAY	Toolbar	This command enables/disables the toolbar display. When it is disabled, the graphic window is larger.
	Softkey	This command enables/disables the softkeys display. When disabled, the graphic window is larger.



?	Help	This command enables the online help of the Machine Plot
	ABOUT MACHINE PLOT	This command displays the Machine Plot release

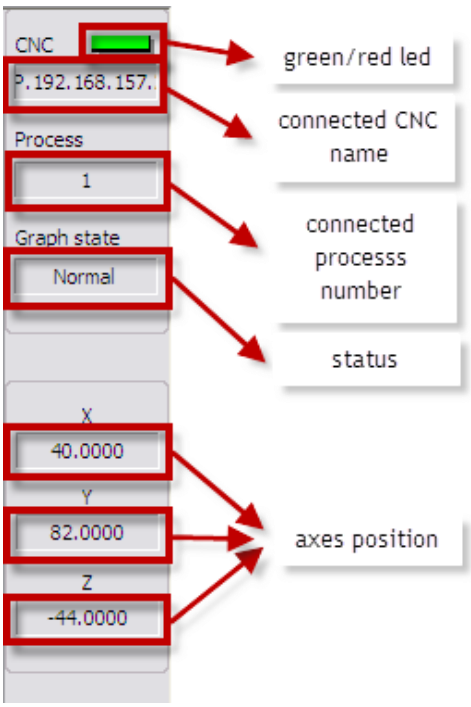
The same commands are also available in the lower softkeys menu

Some more frequently used commands are also present in the button bar



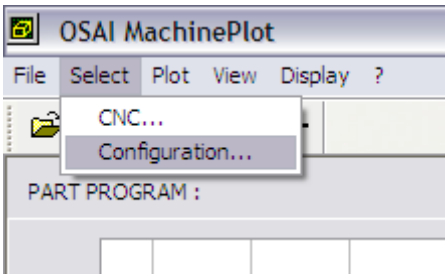
In the vertical bar on the right hand side, the following information is displayed:

- the led identifies the connection status with the CNC: if the CNC is connected, the led is green, otherwise it is red
- name of the connected CNC
- number of the connected process
- status of the graphic (NORMAL or ZOOMED)
- axes positions: the axes positions are displayed during acquisition or the abscissa/ordinate position of the cursor on the video when DIMENSIONING

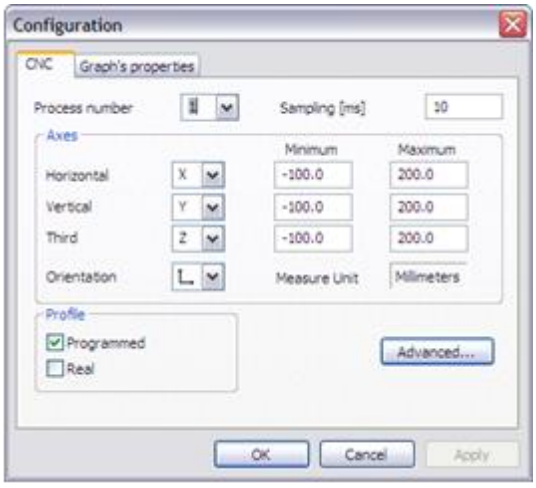


11.3 Machine Plot Configuration

In the toolbar select “SELECT” and then “CONFIGURATION”

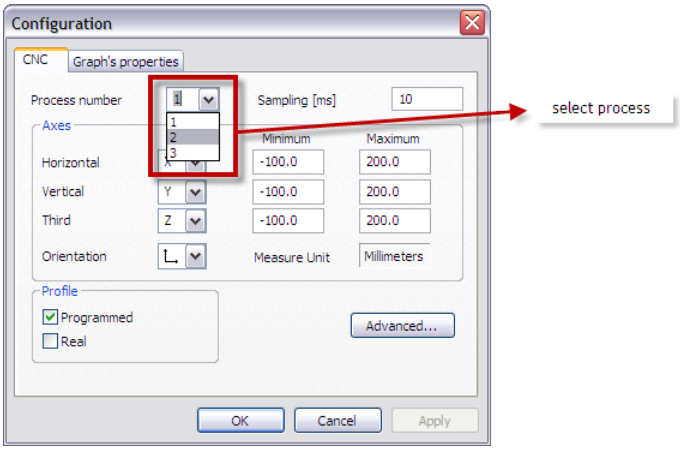


The “CONFIGURATION” window is displayed that allows configuration of the Machine Plot graphic interface.

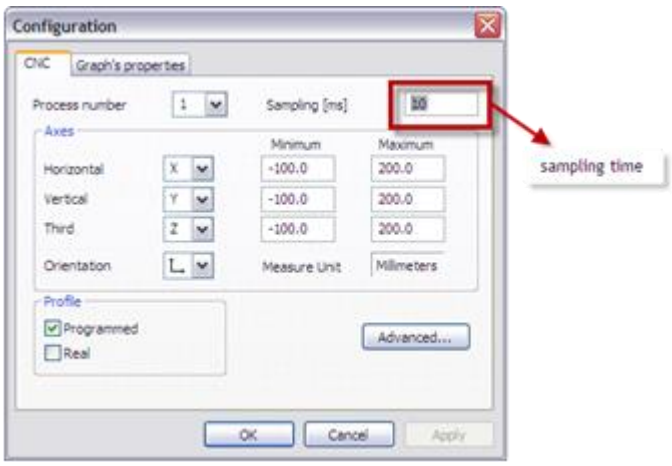


By selecting the CNC tab, the following parameters can be set:

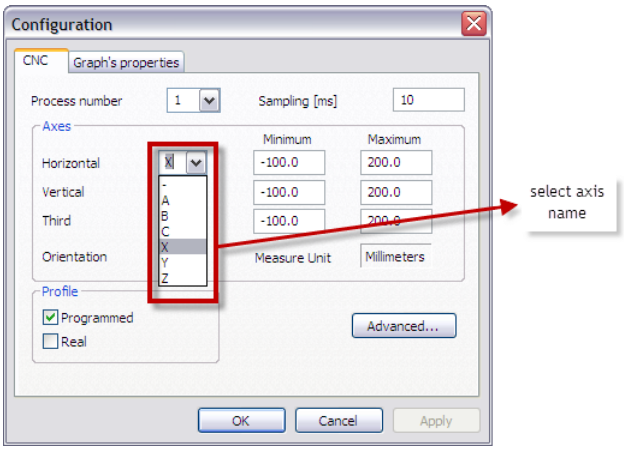
- the number of the desired process: this choice can be made by using the pull-down menu which offers all the processes configured in the CNC. The process on which the program to be analysed must of course be selected.



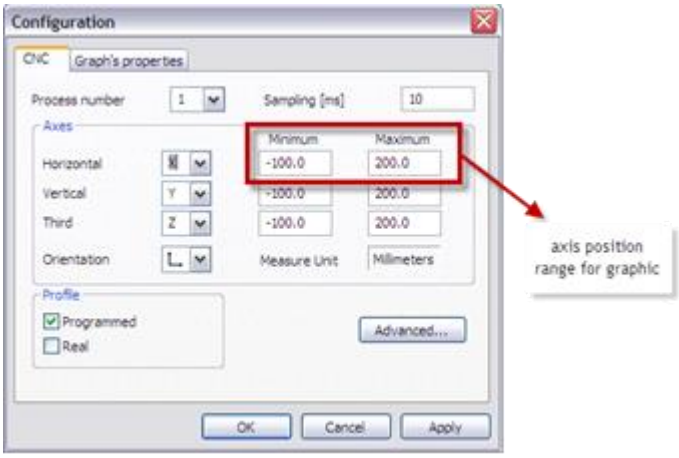
- the position sampling tick in milliseconds: the value set must be a multiple of the system clock, otherwise it displays an error message



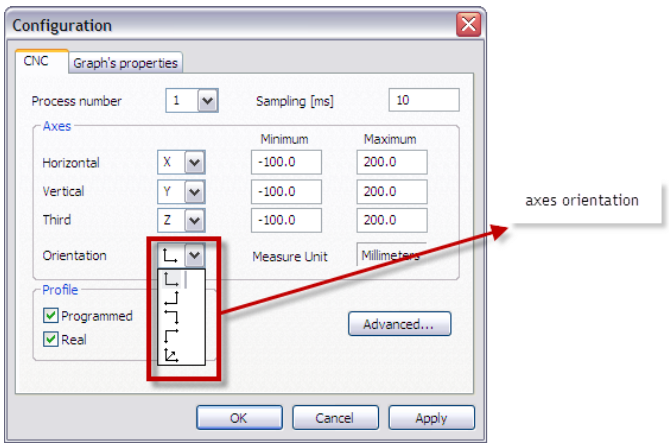
- each one of the three axes (named HORIZONTAL, VERTICAL and THIRD) of the Cartesian reference system of the graphics, must be associated with a physical axis which can be selected using the pull-down menu, from those available (those configured in selected the process)



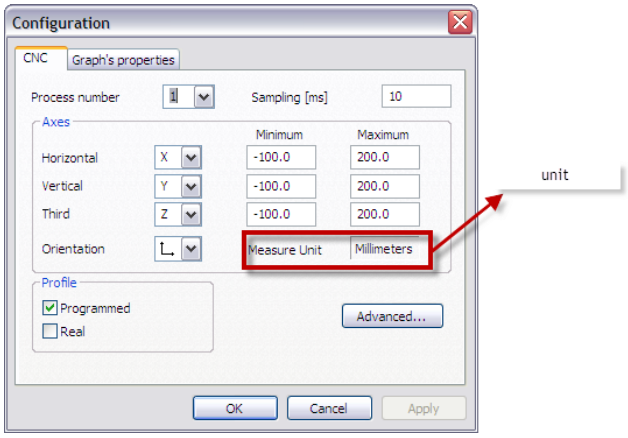
- for each axis of the Cartesian system the display range must be defined by setting a minimum and maximum value



- the direction in which the axes will be represented in the Cartesian axis system can be selected (only for 2D display) via a drop-down menu. From here either the 2D view, which displays only the first two axes HORIZONTAL and VERTICAL, or a 3D display which displays a set of three Cartesian axes may be selected.

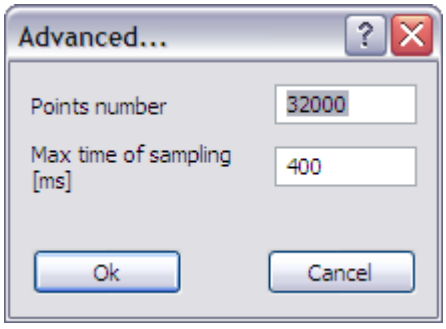


- the configured measurement unit is displayed in a dedicated field



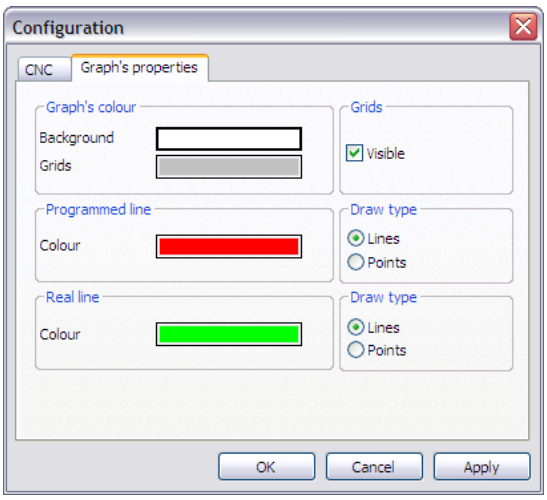
- using the “checks”, select whether to display the theoretical profile (calculated axes position) or also the actual one (real axes position)

- if the "ADVANCED” button is pressed the “ADVANCED” window opens , where the maximum number of points acquired and the maximum sampling clock value can be defined.

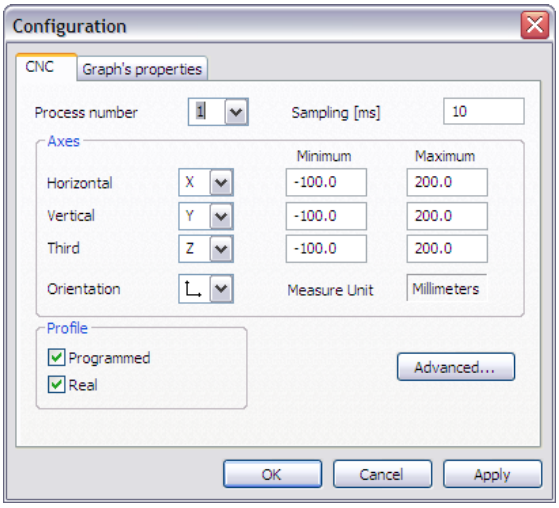


Select the tab “GRAPH’S PROPERTIES” to:

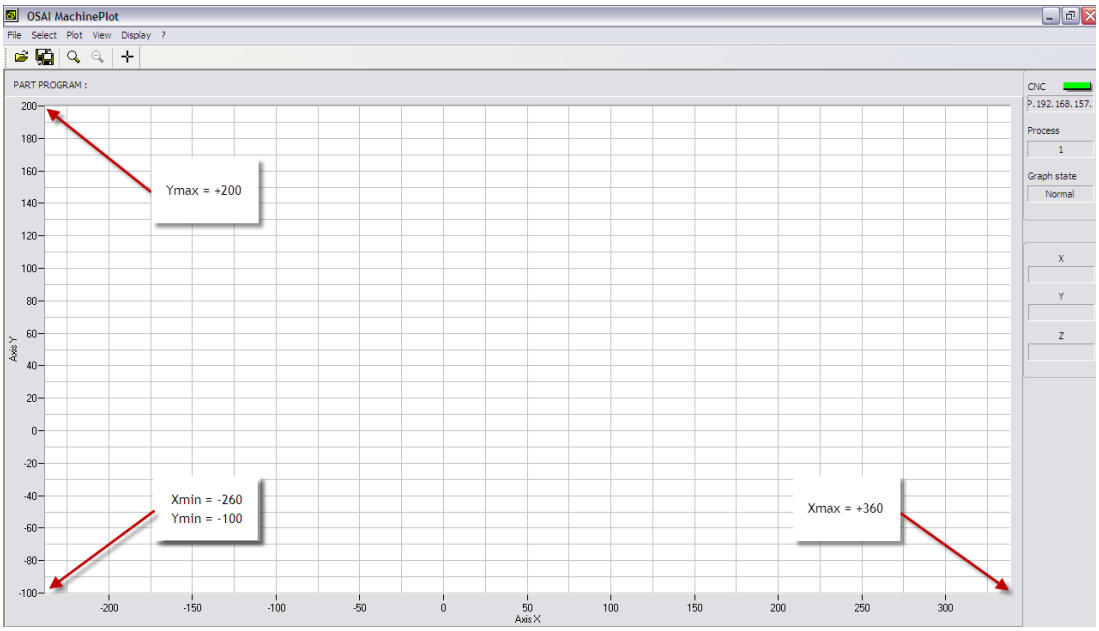
- select the background colour of the graphical video and of the grid
- enable/disable the grid display
- select the colour of the theoretical and real path
- select the display of the path as a dotted or continuous line



For example, if the parameters are set as described in the picture below

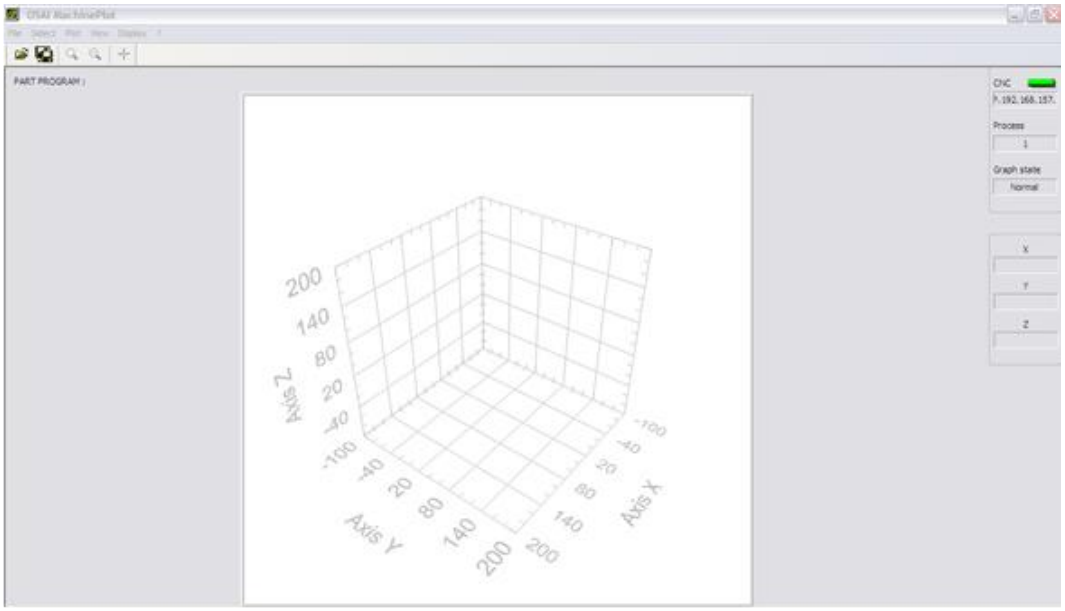


after confirmation with the “OK” button, the following 2D window appears, where the positions of the two Cartesian axes (abscissa and ordinate) adjusted according to the defined maximum & minimum values will be displayed.



Left-click on the graphical area and keep the button pressed while moving the mouse, the displayed profile can be moved within it.

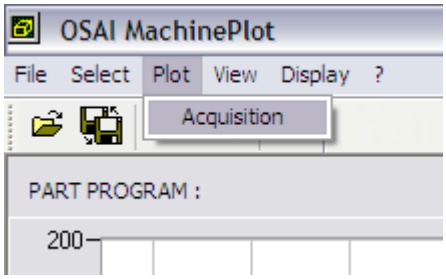
Using the same setting, but selecting the 3D representation, the following window appears



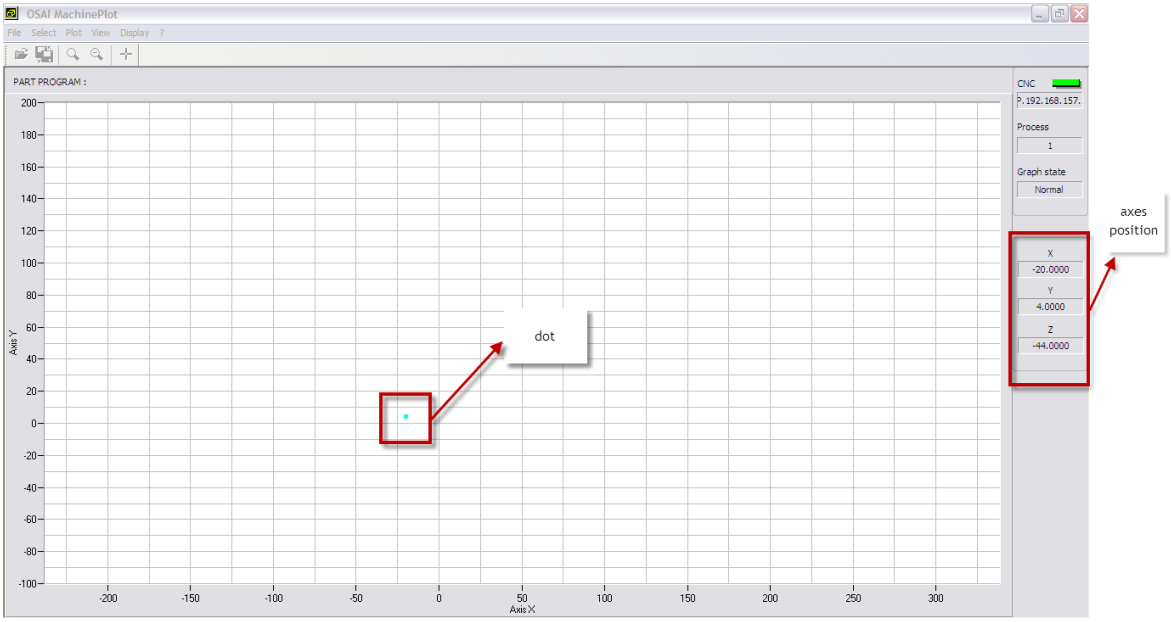
Left-click on the graphical area and by keep the button pressed and moving the mouse the Cartesian coordinate system and all the profile displayed in it can be rotated

11.4 Profile Drawing

To draw the tool path when a program is running, the acquisition must be enabled by selecting “PLOT” from the toolbar followed by “ACQUISITION”.

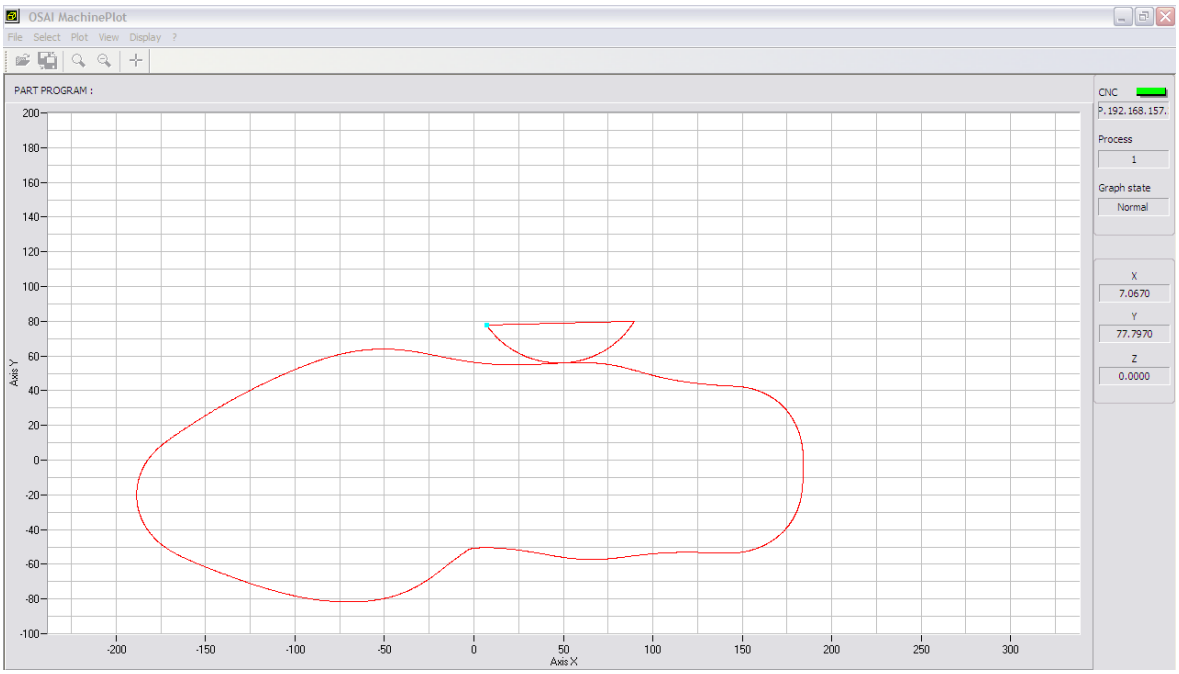


This displays the following window where, in the vertical bar on the right, the current ids of the three configured axes (XYZ) and the position in the XY plane (assuming the 2D display is selected) is represented by a blue point on the screen.



If the blue point, cannot be seen it is probably outside the displayed graphical area: to check this, compare the axes position displayed on the right with the graphical area limits. To move the graphical area to include the blue point, left-click and keep the button pressed and move the mouse to drag the graphical area in the desired direction.

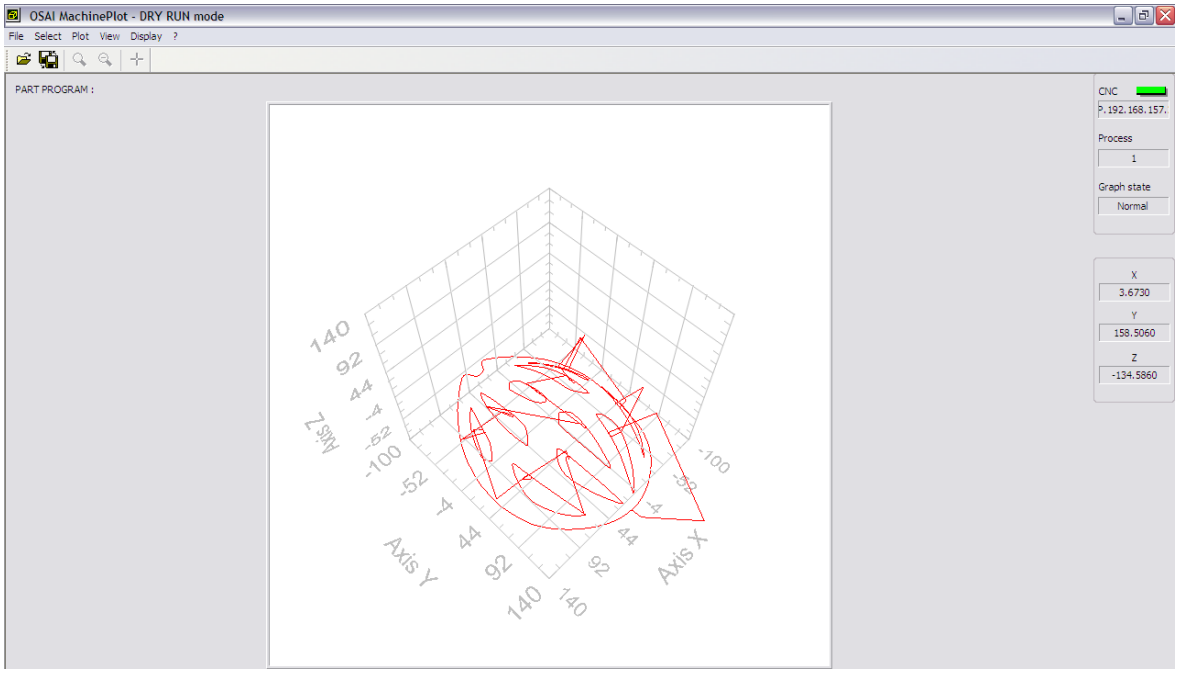
At this point the part program can be executed. The axes position will be updated and will move in the XY plane according to the programmed feeds while its path is traced by a line.



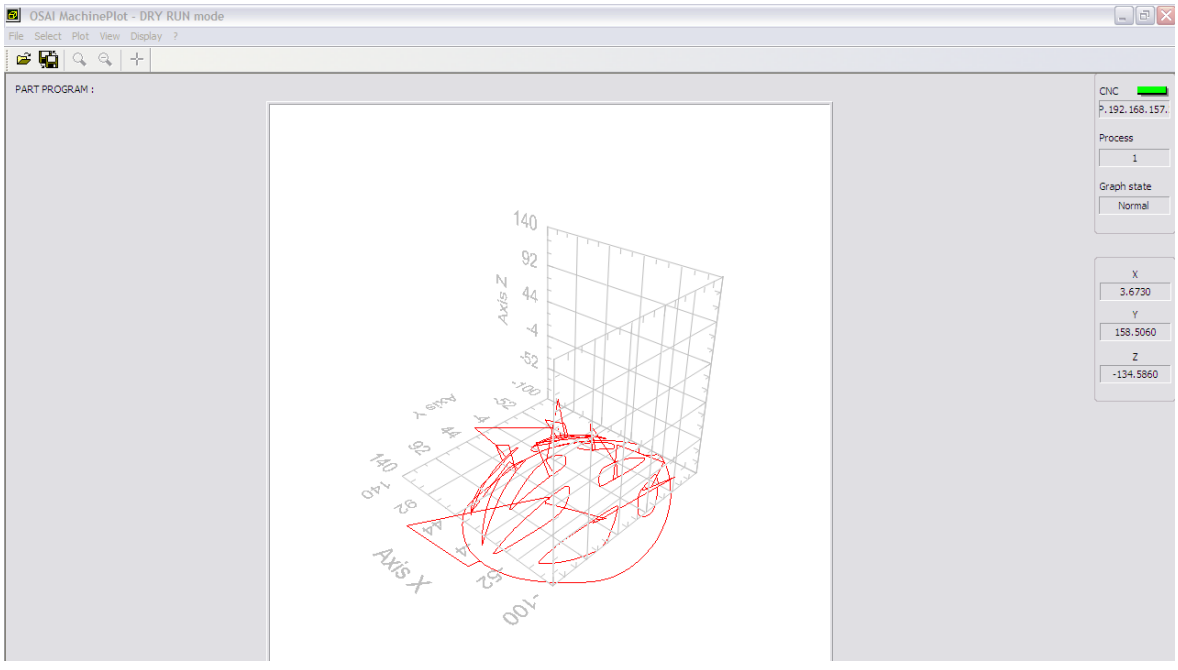
The drawing profile execution can be stopped & started at any time by clicking “PLOT” and then “ACQUISITION” on the toolbar. If the acquisition of the real path has been enabled, when the acquisition stops, the screen displays both theoretical and real profiles.
The points of the density of the profile depends on the configured sampling frequency, the smaller the value, the more points will be displayed and vice versa. The value of this parameter strongly influences the time taken for tracing.

11.5 Drawing a 3D Profile

If a 3D display is selected, a three-dimensional display of the profile is shown, as for example, in the figure below.



Left-click in the graphical area and keep the button pressed while moving the mouse and, the figure can be rotated in all directions.



However, the 3D display has the following limitations:

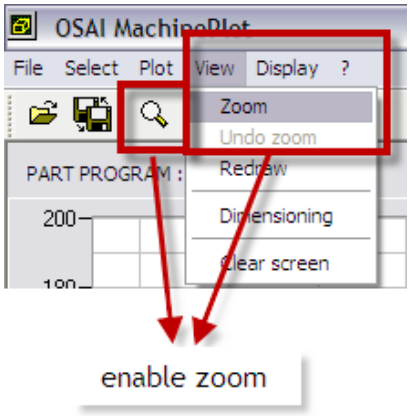
- the real profile cannot be displayed: only the theoretical path is represented
- the ZOOM function is not available
- the DIMENSIONING function is not available

After the acquisition of a profile in 3D mode the 2D setting can be displayed and the same profile may be projected on the XY plane or the YZ plane.

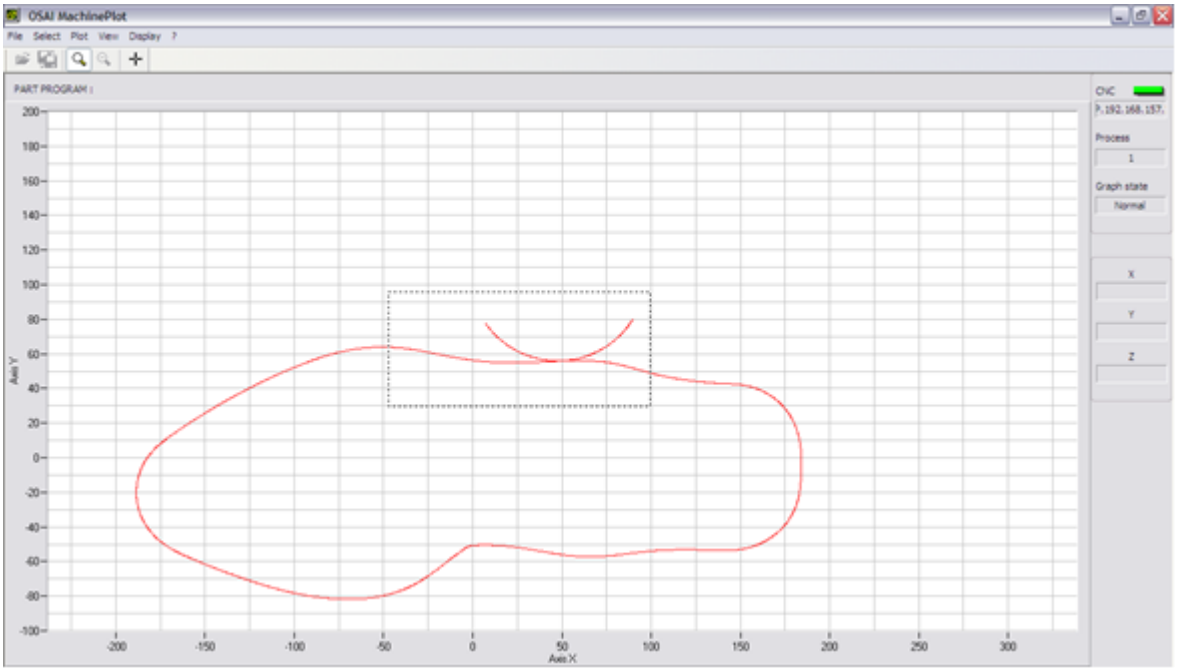
11.6 Checking the Profile

When acquiring the points, only the theoretical profile is displayed, even if the acquisition is enabled also for the real profile. Only when the acquisition is complete is the real profile displayed on the screen. At this point, there are several instruments available to analyse the displayed data.

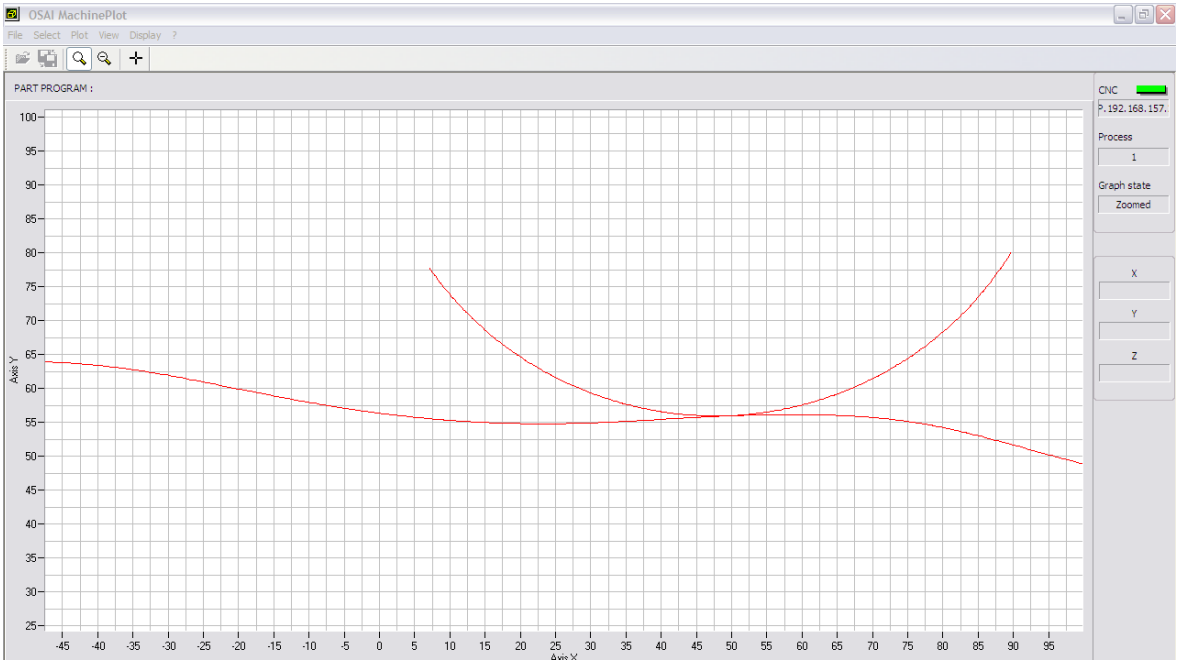
The theoretical and real paths will usually overlap so, to see the differences, the ZOOM function can be used. ZOOM can be enabled by selecting “VIEW” and then “ZOOM” from the toolbar or by pressing the dedicated button.



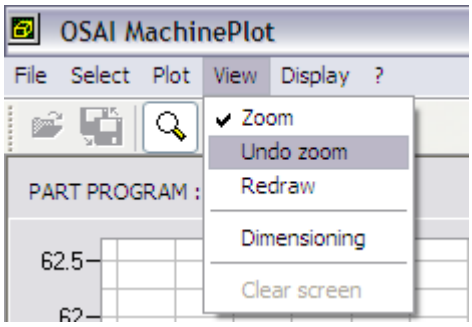
Using drag & drop with the mouse, select the desired part of the profile that with the dashed rectangle.

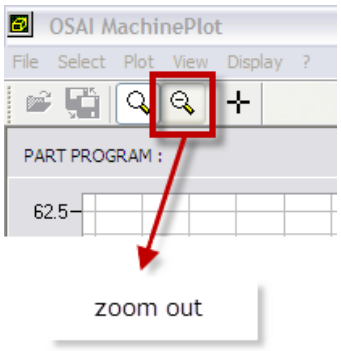


When the mouse button is released , the selected area is displayed full screen. The status bar on the right of the graph changes to ZOOMED. The extremes of the abscissa and ordinate axes are updated accordingly.

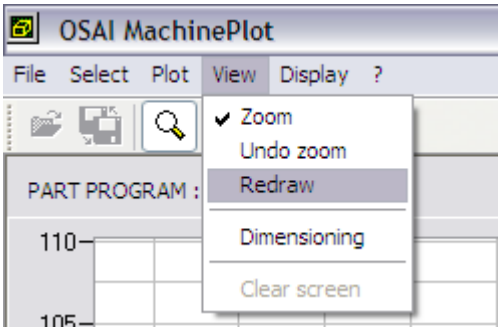


It is possible to zoom in and out to small areas until the Machine Plot prevents. The zoom can be undone step-by-step by selecting “VIEW” from the toolbar and then “UNDO ZOOM” or by using the dedicated button.





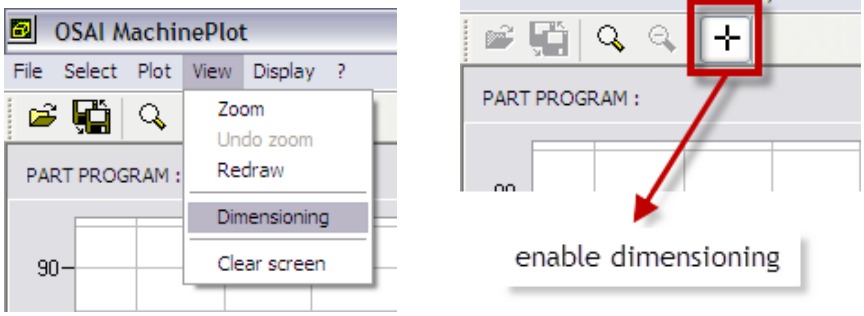
ZOOM can also be reset by selecting “VIEW” from the toolbar and then “REDRAW”, restoring the original dimension display.



The displayed window can be panned by using the mouse: click in a zone of the graphic and keeping button pressed move the mouse to move the window, then release the button. This function is enabled only if the ZOOM is disabled.

On the profile traced on the video it is possible to precisely detect the positions of a single point: the

accuracy of this analysis depends on the current zoom level but can be close to the values of sampling interpolator (depending on the tick sampling configured).
This can be done by using the DIMENSIONING function that can be enabled selecting by “VIEW” from the toolbar and then “DIMENSIONING” or by pressing the dedicated button.



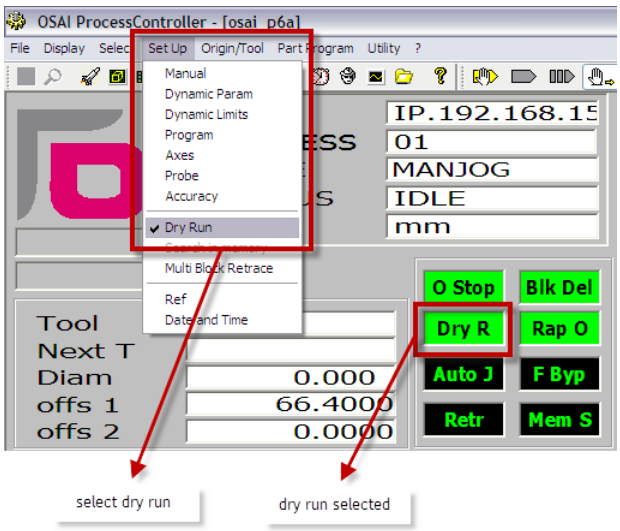
The screen displays a cross on the profile: this cursor can be moved along the profile by moving the mouse.

When moving the cross, the positions displayed in the vertical bar on the right will change representing the position of the cross (theoretical or real) and therefore the position of the axes.

In the case where both the theoretical and the real profile are displayed, by using DIMENSIONING the differences between the two profiles can be measured. This corresponds with very good approximation to the errors encountered on the part.
With this tool a pre-analysis of the profile quality can be determined by performing a test of the program (i.e., moving the axes in the air, without the material to be processed). By analysing the profile, it is possible to check if the real profile, especially in critical areas (edges, joints, ...) satisfies the tolerance requirements and also if there are any mechanical problems (such as jerks or vibration).

11.7 Drawing the disconnected axes profile (DRY RUN)

To plot a profile without moving the axes, the DRY RUN mode must be enabled. Select “SETUP” from the toolbar and then “check” DRY RUN (the DRY RUN status is enabled and indicated by the dedicated led).



The functionality and the usage of the Machine Plot is the same as that described in the previous pages, except that only the theoretical profile is displayed, (because the axes do not physically move).

Normally the profile is executed with the feed defined in the program: if instead the working feed and/or the rapid overrides are reduced to zero, the profile (and consequently the tracking in Machine Plot) runs at the maximum speed of calculation.
This allows a “quick” verification of the profile, also in case of programs having big dimensions, to ensure there are no incorrect movements.

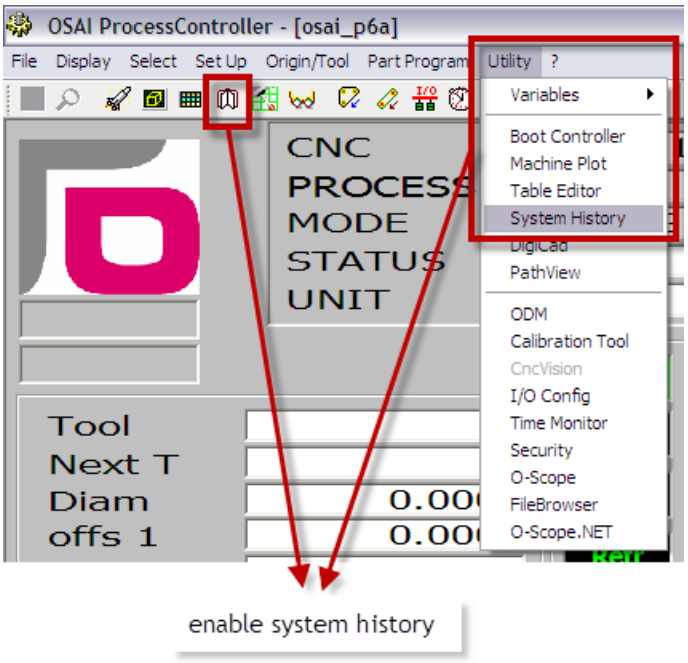
12. System History

12.1 General

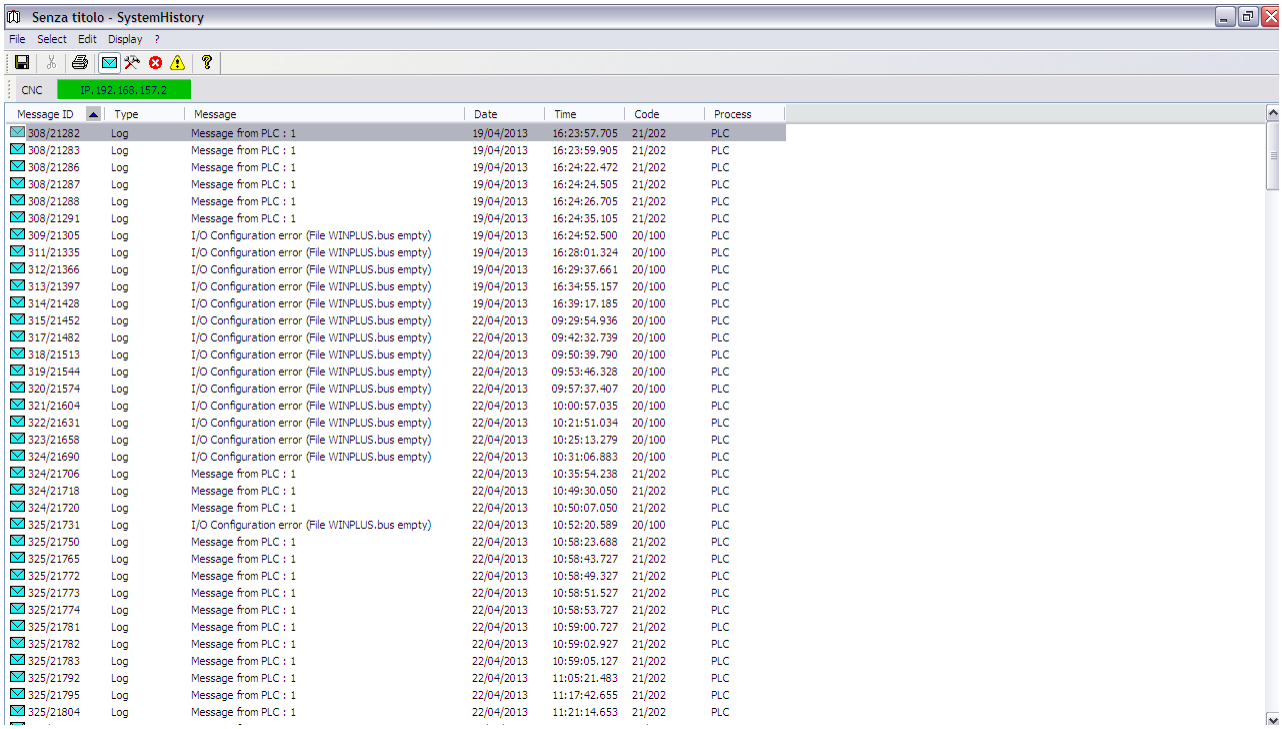
When a warning or emergency message is displayed on the CNC screen, it is also stored in a file of the CNC. Even the messages coming from the logic developed by MTB may possibly (if provided by MTB) be stored in the same file. Up to 300 messages can be stored for each type: when this limit is reached, for each new message that is added to the list, the oldest one is deleted. The messages are stored in chronological order, they are not lost after the CNC is turned off and are immediately available via the System History ambient. In case of necessity, this allows those who have to evaluate the reasons for a machine downtime, to reconstruct what happened (reading the list of messages with all relevant information) and get back to the causes.

12.2 System History Interface

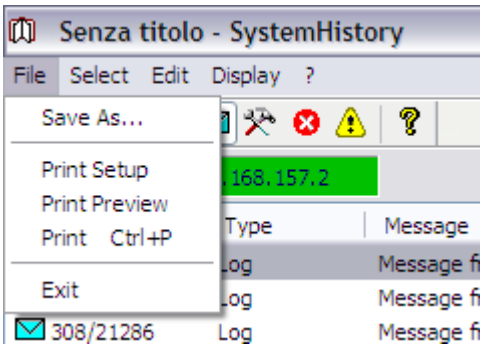
In order to access the System History, select UTILITY and then SYSTEM HISTORY or press the dedicated button in the toolbar.



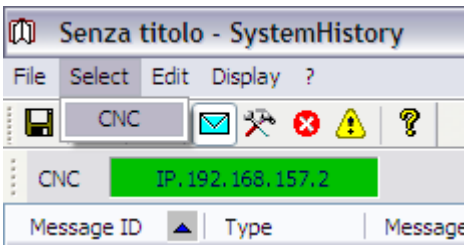
Then, the following window is displayed, listing the LOG type messages (arriving from the system and the PLC) in chronological order, form the older to the newer.


Senza titolo - SystemHistory
File Select Edit Display ?
CNC IP.192.168.157.2
Message ID Type Message Date Time Code Process
308/21282 Log Message from PLC : 1 19/04/2013 16:23:57.705 21/202 PLC
308/21283 Log Message from PLC : 1 19/04/2013 16:23:59.905 21/202 PLC
308/21286 Log Message from PLC : 1 19/04/2013 16:24:22.472 21/202 PLC
308/21287 Log Message from PLC : 1 19/04/2013 16:24:24.505 21/202 PLC
308/21288 Log Message from PLC : 1 19/04/2013 16:24:26.705 21/202 PLC
308/21291 Log Message from PLC : 1 19/04/2013 16:24:35.105 21/202 PLC
309/21305 Log I/O Configuration error (File WINPLUS.bus empty) 19/04/2013 16:24:52.500 20/100 PLC
311/21335 Log I/O Configuration error (File WINPLUS.bus empty) 19/04/2013 16:28:01.324 20/100 PLC
312/21366 Log I/O Configuration error (File WINPLUS.bus empty) 19/04/2013 16:29:37.661 20/100 PLC
313/21397 Log I/O Configuration error (File WINPLUS.bus empty) 19/04/2013 16:34:55.157 20/100 PLC
314/21428 Log I/O Configuration error (File WINPLUS.bus empty) 19/04/2013 16:39:17.185 20/100 PLC
315/21452 Log I/O Configuration error (File WINPLUS.bus empty) 22/04/2013 09:29:54.936 20/100 PLC
317/21482 Log I/O Configuration error (File WINPLUS.bus empty) 22/04/2013 09:42:32.739 20/100 PLC
318/21513 Log I/O Configuration error (File WINPLUS.bus empty) 22/04/2013 09:50:39.790 20/100 PLC
319/21544 Log I/O Configuration error (File WINPLUS.bus empty) 22/04/2013 09:53:46.328 20/100 PLC
320/21574 Log I/O Configuration error (File WINPLUS.bus empty) 22/04/2013 09:57:37.407 20/100 PLC
321/21604 Log I/O Configuration error (File WINPLUS.bus empty) 22/04/2013 10:00:57.035 20/100 PLC
322/21631 Log I/O Configuration error (File WINPLUS.bus empty) 22/04/2013 10:21:51.034 20/100 PLC
323/21658 Log I/O Configuration error (File WINPLUS.bus empty) 22/04/2013 10:25:13.279 20/100 PLC
324/21690 Log I/O Configuration error (File WINPLUS.bus empty) 22/04/2013 10:31:06.883 20/100 PLC
324/21706 Log Message from PLC : 1 22/04/2013 10:35:54.238 21/202 PLC
324/21718 Log Message from PLC : 1 22/04/2013 10:49:30.050 21/202 PLC
324/21720 Log Message from PLC : 1 22/04/2013 10:50:07.050 21/202 PLC
325/21731 Log I/O Configuration error (File WINPLUS.bus empty) 22/04/2013 10:52:20.589 20/100 PLC
325/21750 Log Message from PLC : 1 22/04/2013 10:58:23.688 21/202 PLC
325/21765 Log Message from PLC : 1 22/04/2013 10:58:43.727 21/202 PLC
325/21772 Log Message from PLC : 1 22/04/2013 10:58:49.327 21/202 PLC
325/21773 Log Message from PLC : 1 22/04/2013 10:58:51.527 21/202 PLC
325/21774 Log Message from PLC : 1 22/04/2013 10:58:53.727 21/202 PLC
325/21781 Log Message from PLC : 1 22/04/2013 10:59:00.727 21/202 PLC
325/21782 Log Message from PLC : 1 22/04/2013 10:59:02.927 21/202 PLC
325/21783 Log Message from PLC : 1 22/04/2013 10:59:05.127 21/202 PLC
325/21792 Log Message from PLC : 1 22/04/2013 11:05:21.483 21/202 PLC
325/21795 Log Message from PLC : 1 22/04/2013 11:17:42.655 21/202 PLC
325/21804 Log Message from PLC : 1 22/04/2013 11:21:14.653 21/202 PLC

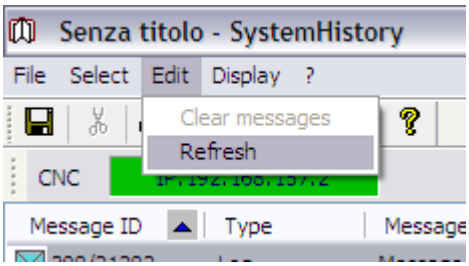
The toolbar contains the following commands



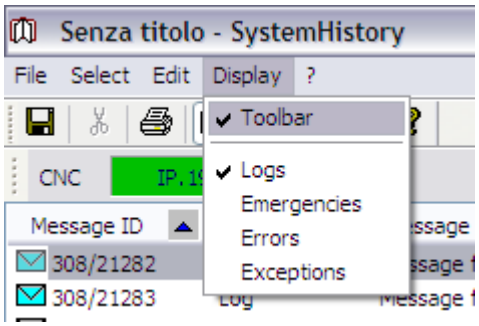
FILE	Save as …	This command saves the list of messages on an ASCII type file. The path and the name of the file are required.
	Print Setup	This command selects the printer and the print options.
	Print preview	This command enables the print preview
	Print	This command launches the print
	Exit	This command allows the exit from the System History ambient.



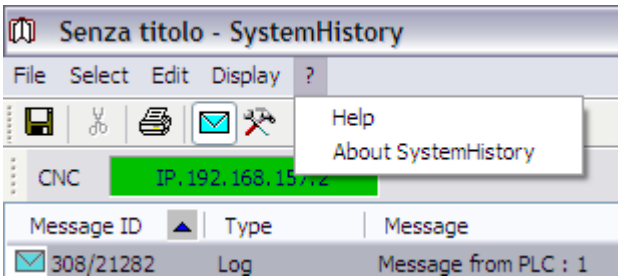
SELECT	CNC	This command defines the name of the CNC to be used for the connection



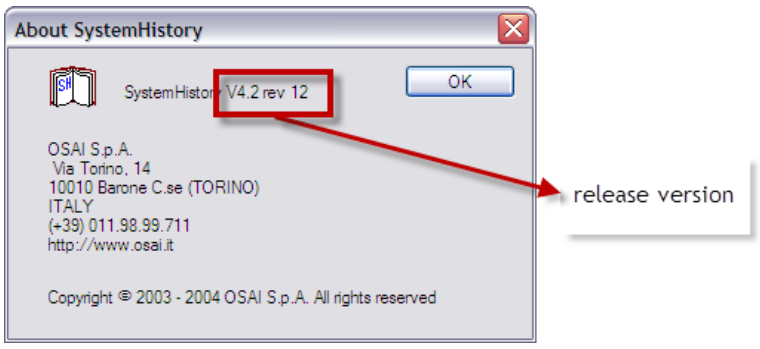
EDIT	Clear message	This command deletes the messages in the list. It is enabled only if the CNC is powered in EMERGENCY MODE.
	Refresh	This command refreshes the list of messages



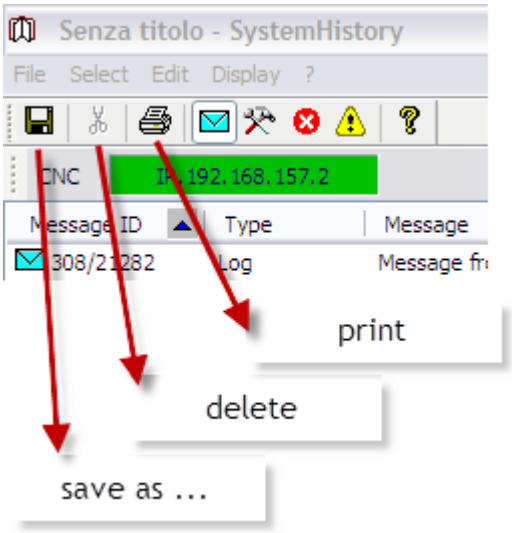
DISPLAY	Toolbar	This command enables/disables the toolbar display.
	Logs	This command enables/disables the display of the LOG type messages. The same function is assigned to the dedicated button.
	Emergencies	This command enables/disables the display of the EMERGENCY type messages. The same function is assigned to the dedicated button.
	Errors	This command enables/disables the display of the ERROR type. The same function is assigned to the dedicated button.
	Exceptions	This command enables/disables the display of the EXECPTION type. The same function is assigned to the dedicated button.

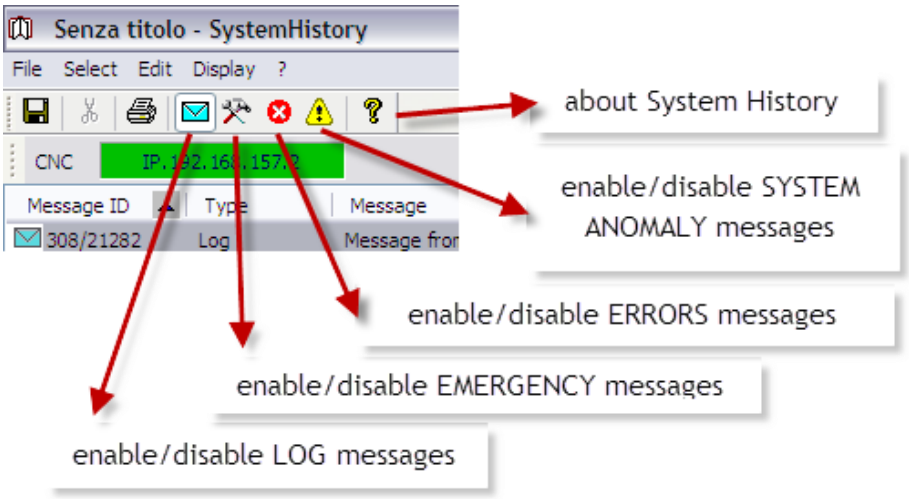


?	Help	This command launches the ON-LINE HELP of the System History ambient
	About System History	This command displays the System History release



Some more frequently used commands are also present in the button bar



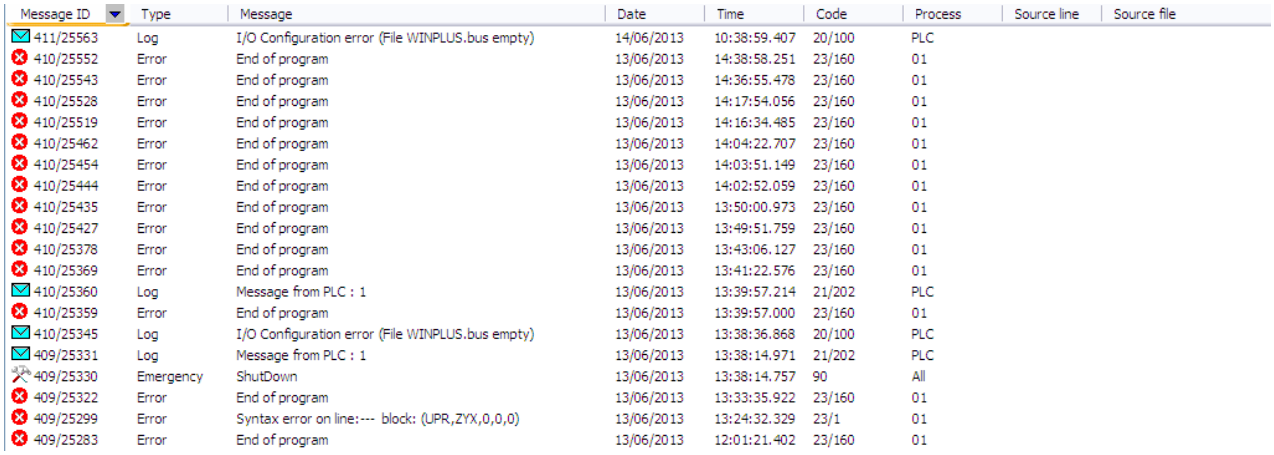


The available messages are grouped in four types:

- LOG: messages arriving from the system and the PLC
- EMERGENCY: emergency messages (servo error, overtravel,…)
- ERRORS: ISO programming error messages
- SYSTEM ANONALIES: messages generated in case of serious anomalies of the CNC. These messages have to be signalled to the Prima Electro technical assistance.

Only when the type is enables, the relative massages appear in the list: this allows the selection to display only the messages of interest.

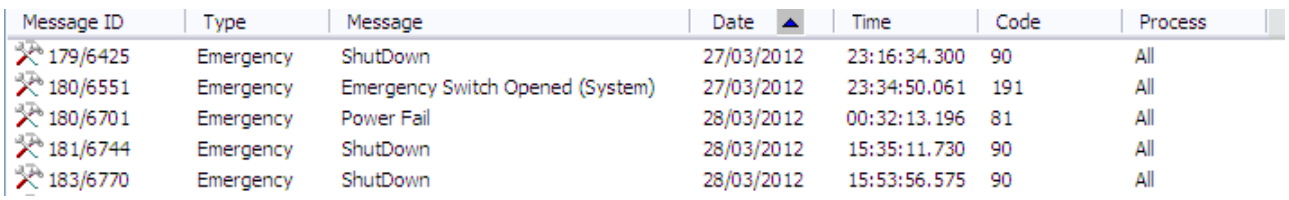
The list of messages is divided in columns listing the following information:



Message ID	Type	Message	Date	Time	Code	Process	Source line	Source file
411/25563	Log	I/O Configuration error (File WINPLUS.bus empty)	14/06/2013	10:38:59.407	20/100	PLC		
410/25552	Error	End of program	13/06/2013	14:38:58.251	23/160	01		
410/25543	Error	End of program	13/06/2013	14:36:55.478	23/160	01		
410/25528	Error	End of program	13/06/2013	14:17:54.056	23/160	01		
410/25519	Error	End of program	13/06/2013	14:16:34.485	23/160	01		
410/25462	Error	End of program	13/06/2013	14:04:22.707	23/160	01		
410/25454	Error	End of program	13/06/2013	14:03:51.149	23/160	01		
410/25444	Error	End of program	13/06/2013	14:02:52.059	23/160	01		
410/25435	Error	End of program	13/06/2013	13:50:00.973	23/160	01		
410/25427	Error	End of program	13/06/2013	13:49:51.759	23/160	01		
410/25378	Error	End of program	13/06/2013	13:43:06.127	23/160	01		
410/25369	Error	End of program	13/06/2013	13:41:22.576	23/160	01		
410/25360	Log	Message from PLC : 1	13/06/2013	13:39:57.214	21/202	PLC		
410/25359	Error	End of program	13/06/2013	13:39:57.000	23/160	01		
410/25345	Log	I/O Configuration error (File WINPLUS.bus empty)	13/06/2013	13:38:36.868	20/100	PLC		
409/25331	Log	Message from PLC : 1	13/06/2013	13:38:14.971	21/202	PLC		
409/25330	Emergency	ShutDown	13/06/2013	13:38:14.757	90	All		
409/25322	Error	End of program	13/06/2013	13:33:35.922	23/160	01		
409/25299	Error	Syntax error on line:--- block: (UPR,ZYX,0,0,0)	13/06/2013	13:24:32.329	23/1	01		
409/25283	Error	End of program	13/06/2013	12:01:21.402	23/160	01		

Message ID	This field contains the icon identifying the message type and the identification number within the message
Type	This field contains the message type
Message	This field contains the extended message
Date	This field contains the data when the message was generated
Time	This field contains the time when the message was generated
Code	This field contains the error code identifying the message in the user manuals
Process	This field contains the ambient that generated the message
Source line **Source file**	Only in case of SYSTEM ANOMALY message type, these two fields provide additional information to identify the origin of the anomaly. These communications have to be told to the Prima Electro technical staff.

Left clicking on the label for each column, it is possible to invert the messages order. For example, if the messages are displayed in chronological ascending order (from the oldest to the newest)),



Message ID	Type	Message	Date	Time	Code	Process
179/6425	Emergency	ShutDown	27/03/2012	23:16:34.300	90	All
180/6551	Emergency	Emergency Switch Opened (System)	27/03/2012	23:34:50.061	191	All
180/6701	Emergency	Power Fail	28/03/2012	00:32:13.196	81	All
181/6744	Emergency	ShutDown	28/03/2012	15:35:11.730	90	All
183/6770	Emergency	ShutDown	28/03/2012	15:53:56.575	90	All

instead of scroll the list to check the most recent messages, you can click on the DATE label to invert the order (from the newest to the oldest).

Message ID	Type	Message	Date	Time	Code	Process
411/25563	Log	I/O Configuration error (File WINPLU...	14/06/2013	10:38:59.407	20/100	PLC
410/25552	Error	End of program	13/06/2013	14:38:58.251	23/160	01
410/25543	Error	End of program	13/06/2013	14:36:55.478	23/160	01
410/25528	Error	End of program	13/06/2013	14:17:54.056	23/160	01
410/25519	Error	End of program	13/06/2013	14:16:34.485	23/160	01

With the same technique it is possible to order the message display by number by type, by code and so on.

13. Appendix A

13.1 Error Messages

```
// Common section
[COMMON]
LOGMSG_LABEL = Log
ERROR_LABEL = Error
EMERGENCY_LABEL = Emergency
EXCEPTION_LABEL = Exception
PROCESS = Process
ALL_PROCESS = All
WARNING_LABEL = Warning

// ---- Log messages ----
//  0/x  Log 4Control
// 17/x  Log IPC
// 18/x  Log Console
// 19/x  Log Emergency
// 20/x  Log I/O
// 21/x  Log PLC
// 22/x  Log Motion
// 23/x  Log CNC
// 24/x  Log OS3
// 25/x  Log Table
// 26/x  Log Geometry
// 27/x  Log Splines
// 31/x  Log Loader
// 32/x  Log ServoLoop
// 50/x  Log Security
// 53/x  Log CanOpen
// 54/x  Log EtherCAT
// 61/x  Log  _3DInterface

[LOGMSG]
19/1 = System Exception %1 in %2
19/2 = System Anomaly %1 in %2

20/100 = I/O Configuration error (%1)
20/101 = I/O CANOPEN Configuration error (%1) board %2 Node %3
20/102 = I/O OsWire Configuration error (%1) FW version %2 Node %3 Slot %4
20/103 = I/O Local Configuration error (%1) Node %2

21/200 = PLC Project %1 loaded
21/201 = PLC Project NOT loaded
21/202 = Message from PLC : %1

23/185 = "AliasGlobal.txt" - defined (%1) aliases - no errors
23/186 = "AliasGlobal.txt" - defined (%1) and ignored (%2) aliases - duplicated alias at line %3
```

23/187 = "AliasGlobal.txt" - defined (%1) and ignored (%2) aliases - too long or incorrectly defined alias at line %3
23/188 = "AliasGlobal.txt" - defined (%1) and ignored (%2) aliases - too many aliases defined at line %3
23/193 = "AliasGlobal.txt" - file not found - no aliases defined
23/801 = NCUSER warning - (%1)

31/1 = Boot IPC
31/2 = Boot Services
31/3 = Boot PLC
31/4 = Boot Axes Servo
31/5 = Boot I/O
31/6 = Boot Motion
31/7 = Boot CNC
31/8 = Boot Serial Line
31/9 = Boot Teach Pendant
31/10 = Shared memory definition
31/11 = Shared memory released
31/12 = Broadcasting definition
31/13 = Broadcasting released
31/20 = System is ON
31/21 = System is OFF
31/22 = Reading System Configuration
31/23 = System Configuration released
31/24 = Activated Configuration is %1
31/25 = Release %1
31/30 = Init Retain Memory
31/31 = Error loading a DLL : %1
31/32 = Software initialisation error : %1
31/33 = Hardware initialisation error : %1
31/34 = Loading field protocols
31/35 = Boot CNCsupport library

32/300 = Servo notice : %1
32/301 = Not possible to verify drive's congruence
32/302 = Verifying drives parameters ...
32/303 = Axes Boot Fatal Error
32/304 = Drives parameters not congruent !
32/305 = Drives parameterization not valid !
32/306 = Board 0 not ready (cause : %1, %2)
32/307 = Board 1 not ready (cause : %1, %2)
32/308 = Board 2 not ready (cause : %1, %2)
32/309 = Board 3 not ready (cause : %1, %2)
32/310 = Drive CANOpen need reboot!
32/311 = CANOpen Configuration : %1 (%2)
32/312 = Mechatrolink notice : %1 (axis ID %2)

50/256 = System configuration error (%1)

54/128 = Error during EtherCAT bus scan (Int err : %1)

```
// ---- Error messages ----
//  0/x  Error 4Control
// 17/x  Error IPC
// 18/x  Error Console
// 19/x  Error Emergency
// 20/x  Error I/O
// 21/x  Error PLC
// 22/x  Error Motion
// 23/x  Error CNC
// 24/x  Error OS3
// 25/x  Error Table
// 26/x  Error Geometry
// 27/x  Error Splines
// 31/x  Error Loader
// 32/x  Error Servoloop
// 33/x  Error OS3 Drive
// 34/x  Error SERCOS Drive
// 35/x  Error CAN Drive
// 36/x  Error SENSOR (Servo)
// 37/x  Error EtherCAT (Servo)
// 38/x  Error Mechatrolink Drive
// 49/x  Error Serial Line
// 50/x  Error Security
// 51/x  Error SERCOS DLL
// 52/x  Error FieldBus DLL
// 53/x  Error CANopen DLL
// 54/x  Error EtherCAT DLL
// 56/x  Error XML DLL
// 61/x  Error Mechatrolink DLL

[ERROR]
0/32 = Illegal Opcode
0/33 = Stack overflow
0/34 = Stack underflow
0/35 = Division by zero
0/36 = Array range
0/37 = String length
0/38 = Null pointer
0/39 = Call Stack overflow
0/40 = Illegal library function/FB call
0/41 = Invalid argument passed to function/FB
0/42 = General mathematic exception
0/43 = Watchdog triggered for task
0/44 = Task execution time overrun
0/45 = UserException
0/46 = Error accessing the VM task image
0/47 = Error activating the VM task timer
0/48 = Error initializing the VM task
0/49 = VM in breakpoint when close debug mode
```

0/50 = ...Free fo User

0/256 = Time Out
0/257 = No connection to OPC server
0/258 = Command already handled
0/259 = Task is in breakpoint
0/260 = Checksum calculation failed
0/261 = Command divided
0/262 = Data truncated
0/263 = No message in message queue
0/264 = Retain memory invalid, executing a cold start
0/265 = Task halted immediately
0/266 = Process still in progress

0/512 = Task initializations failed
0/513 = Invalid task number
0/514 = Invalid command received
0/515 = Invalid data received
0/516 = Invalid object offset
0/517 = Invalid instance object
0/518 = Invalid class ID
0/519 = Data size not valid
0/520 = Message or breakpoint queue full
0/521 = Out of memory
0/522 = Not in debug mode
0/523 = Not logged in
0/524 = No project loaded
0/525 = Error entering a critical section
0/526 = Error activating a VM timer
0/527 = Project conflict. New download required
0/528 = To many simultaneous variables. Buffer overrun
0/529 = Invalid parameter
0/530 = Unable to create IPC queue
0/531 = Unable to create VM timer
0/532 = Unable to receive a message on a IPC queue
0/533 = Unable to send a message on a IPC queue
0/534 = Time Out waiting for VMM
0/535 = Already in Debug Mode. Only one Debug Connection possible
0/536 = Invalid object segment received
0/537 = Error reading from communication device
0/538 = Error writing to communication device
0/539 = Not connected
0/540 = Invalid message received
0/541 = Write access to input segment
0/542 = To much data specified
0/543 = Function/Command not supported
0/544 = Wrong - not fitting - response received
0/545 = Common error message received
0/546 = An invalid message queue was given
0/547 = Unknown 4CONTROL product ID given

0/548 = Unable to execute command
0/549 = An invalid IO layer was given
0/550 = Feature not licensed
0/551 = Invalid license key received
0/552 = Invalid product number received

0/768 = Invalid download domain
0/769 = Invalid version
0/770 = To many tasks specified
0/771 = To many instance objects
0/772 = To many classes (POU's)
0/773 = To many programs per task
0/774 = Code segment full
0/775 = Data segment full
0/776 = To many read/write memory regions
0/777 = Not all expected items received
0/778 = Data segment initializations out of order
0/784 = Error creating a task or thread
0/785 = To many local retain copy segments
0/786 = Temporary buffer is too small
0/787 = Error creating a task primary queue
0/788 = Error creating a task secondary queue
0/789 = Error creating a task Event Flag
0/790 = Error creating windows socket
0/791 = Error creating a semaphore
0/792 = Error getting base timer

0/1024 = Invalid code position
0/1025 = Breakpoint not found in breakpoint list
0/1026 = Breakpoint list full
0/1027 = Task is not in breakpoint state

0/1280 = File library not initialized
0/1281 = To many open files
0/1282 = Invalid file ID used
0/1283 = End of file reached
0/1284 = Error opening a file
0/1285 = Error closing a file
0/1286 = Error reading from file
0/1287 = Error writing to file
0/1288 = Temporary file closed by another process
0/1289 = File does not exist
0/1290 = Error setting file position
0/1291 = Error renaming a file
0/1292 = Error deleting a file
0/1293 = Invalid drive specification
0/1294 = Error changing drive
0/1295 = Error changing directory
0/1296 = Invalid directory name
0/1287 = Invalid file name

0/1298 = File path is not in 8.3 format
0/1299 = File path to long
0/1300 = Invalid file specified

0/1536 = Debug library not initialized
0/1537 = Parameter not valid
0/1538 = Object not found
0/1539 = Debug file invalid formatted
0/1540 = Complex variable has mixed with retentive and non-retentive memory
0/1541 = Communication buffer to small

0/1792 = Common flash error
0/1793 = Error initializing the flash memory
0/1794 = Error writing into the flash memory
0/1795 = Error reading from the flash memory
0/1796 = Error erasing the flash memory
0/1797 = Flash domain does not match

0/2048 = Number of tasks or tasks attributes changed. Online Change not possible
0/2049 = Project changed. Online Change not possible
0/2050 = Invalid code object index received
0/2051 = Invalid data object index received
0/2052 = Invalid data copy list received
0/2053 = Invalid new data object index specified
0/2054 = Invalid old data object index specified
0/2055 = To many changed code objects received
0/2056 = To many changed data objects received
0/2057 = To many entries in data object copy list
0/2058 = To many temporary code objects
0/2059 = To many temporary data objects
0/2060 = Online Change is not supported for retain variables

17/1 = Null Ambient
17/2 = Null Task Id
17/3 = Task already defined
17/4 = Wrong Task priority
17/5 = Wrong Task quantum
17/6 = Too many tasks
17/7 = CreateThread Error
17/8 = Task not defined
17/9 = Task cannot be deleted
17/10 = Task cannot suspend itself
17/20 = DLL used by many processes
17/21 = Error loading a DLL
17/22 = DLL not found
17/23 = Too many DLLs
17/24 = Null DLL Id
17/25 = Wrong DLL Version (function not found)
17/26 = DLL not loaded
17/40 = Null Queue Id

17/41 = Secondary Queue not found
17/42 = Primary Queue not found
17/43 = Too many Queues
17/44 = Queue already defined
17/45 = CreateSemaphore error
17/46 = Queue not defined
17/47 = Too many Default Queues
17/48 = Default Queue not defined
17/49 = Default Queue must have Id = 0
17/50 = Default Queue already defined
17/51 = Broadcasting Queue already defined
17/52 = Too many Broadcasting Queues
17/53 = Broadcasting Queue not defined
17/54 = Too many Queue linked to a Broadcasting Queue
17/55 = Queue not linked to a Broadcasting Queue
17/56 = Queue already linked to a Broadcasting Queue
17/57 = Queue memory full
17/58 = Timeout on Send messages to a Queue (memory full)
17/59 = WaitForSingleObject error (send queue)
17/60 = A Task is Pending on Queue
17/61 = Timeout on Pending on Queue
17/62 = WaitForSingleObject error (pend queue)
17/63 = Empty Queue on Queue Accept
17/64 = Queue cannot be deleted (messages)
17/65 = Primary/Secondary Queue not defined
17/70 = Null Event Id
17/71 = Too many Events
17/72 = Event already defined
17/73 = Event not defined
17/74 = CreateEvent error
17/75 = A Task is Pending on Event
17/76 = Timeout on Pending on Event
17/77 = WaitForSingleObject error (pend event)
17/78 = Broadcasting Event already defined
17/79 = Too many Broadcasting Events
17/80 = Broadcasting Event not defined
17/81 = Event already linked to Broadcasting Event
17/82 = Too many Events linked to Broadcasting Event
17/83 = Event not linked to Broadcasting Event
17/100 = Null Semaphore Id
17/101 = Semaphore already defined
17/102 = Too many Semaphores
17/103 = Semaphore not defined
17/104 = CreateSemaphore error
17/105 = A Task is Pending on Semaphore
17/106 = Timeout Pending on Semaphore
17/107 = WaitForSingleObject error (pend semaphore)
17/108 = Semaphore Overflow
17/120 = Null Mutex Id
17/121 = Mutex already defined

17/122 = Too many Mutexs
17/123 = Mutex not defined
17/124 = CreateMutex error
17/125 = A Task is Pending on Semaphore
17/126 = Timeout Pending on Mutex
17/127 = WaitForSingleObject error (pend mutex)
17/128 = Task nesting error on Mutex
17/140 = Error creating System Timer task
17/141 = KernelIOctrl error
17/142 = LoadIntChain error
17/143 = CreateISTevent error
17/144 = Irqinit error
17/145 = Device not defined
17/146 = IoControl error
17/147 = WaitForSingleObject error (pend device)
17/148 = Timeout Pending on Device
17/149 = Null Device Id
17/150 = Device already defined
17/151 = Too many Devices
17/152 = Wrong IRQ
17/153 = Request not available for System Devices
17/154 = A Task is Pending on Device
17/155 = System Timer error
17/156 = GIISR_INFO parameter error
17/160 = Null Timer Routine Id
17/161 = Timer Routine not defined
17/162 = Timer Routine already defined
17/163 = CreateEvent error
17/164 = Too many Timer Routines
17/165 = Wrong Timer Routine Group
17/166 = Wrong timer Period not congruent with the Group
17/167 = System Timer defined
17/168 = A Task is Pending on Timer
17/169 = Timeout Pending on Timer
17/170 = WaitForSingleObject error (pend timer)
17/180 = Null Memory Id
17/181 = Memory already defined
17/182 = Too many memories
17/183 = Memory not defined
17/184 = Memory in use
17/185 = CreateFileMapping error
17/186 = MapViewOfFile error
17/187 = VirtualAlloc error
17/188 = VirtualCopy error
17/189 = Memry not allocated
17/190 = Too many Processes
17/200 = Error allocating Queue memory
17/201 = Error accessing Queue memory
17/202 = Error creating Queue memory Event
17/210 = Wrong Debug Item type

17/220 = Registry RegCreateKeyEx error
17/221 = Registry RegQueryValueEx error
17/222 = Error creating Timeout Event
17/224 = Registry RegSetValueEx error
17/225 = Registry RegFlushKey error
17/226 = Registry RegDeleteValue error
17/230 = Too many monitoring items
17/231 = Wrong Monitoring function
17/232 = Undefined monitoring item
17/233 = Monitoring item already defined
17/250 = IPC Driver Activation Error
17/251 = IPC Drive Open Error
17/252 = Illegal Board Number
17/253 = System Tick Value Error
17/254 = CEWin Timer Error
17/255 = I/O Timer (10 ms) Value Eroor
17/256 = Wrong Syncode OPENcontrol message management
17/257 = Wrong Message Len OPENcontrol message management
17/258 = One Shot Timer Value Error
17/259 = Servo Time Value Error
17/260 = Shift Time Value Error
17/261 = Watch Dog Already Actived
17/262 = Watch Dog Time Value Error
17/263 = Led Number Error
17/264 = Illegal Memory Access from Driver
17/270 = Axis Board not Found
17/271 = Axis Board initialisation error
17/272 = Acknoledge variable NOT writable
17/273 = Firmware Binary file not Found
17/274 = Board Firmware Corrupted
17/275 = Error during FPGA upload
17/276 = Error during FPGA download
17/277 = Error during FPGA erase
17/278 = Upload/Download FPGA not allowed on the installed board
17/500 = Access Violation Exception
17/501 = Array Bounds Exception
17/502 = Breakpoint Exception
17/503 = Datatype Misalignment Exception
17/504 = Denormal Operand Exception
17/505 = Divide by Zero Exception
17/506 = Inexact Result Exception
17/507 = Invalid Operation Exception
17/508 = Overflow Exception
17/509 = Stack Check Exception
17/510 = Underflow Exception
17/511 = Illegal Instruction Exception
17/512 = Page Error Exception
17/513 = Integer Divide By Zero Exception
17/514 = Integer Overflow Exception
17/515 = Invalid Disposition exception

17/516 = Non Continuable Exception
17/517 = Priviledged Instruction Exception
17/518 = Single Step Exception
17/519 = Stack Overflow Exception

18/1 = Too many consoled defined
18/2 = Console already defined
18/3 = Console not defined on I/O Bus
18/4 = Process not defined
18/5 = Too many consoles for a process
18/6 = Console already in use on the requiring process
18/7 = Console not assigned to the removing process
18/8 = Console cannot be removed because assigned to a process
18/9 = Wrong selector Id
18/10 = Wrong number of steps
18/11 = Wrong step value
18/12 = Wrong step value into the array sequence
18/13 = Wrong array direction value
18/14 = I/O Bus without consoles or Bus not defined
18/15 = Unknown Bus

19/1 = Memory allocation error
19/2 = Emergency memory full
19/3 = Watch Dog Initialisation Error
19/4 = Watch Dog Stop Error

20/1 = I/O Setup file not found
20/2 = Wrong setup command line
20/3 = Wrong Node offset for inputs
20/4 = Wrong Node offset for outputs
20/5 = Wrong Slot offset for inputs
20/6 = Wrong Slot offset for outputs
20/7 = Wrong 2nd Slot offset for inputs
20/8 = Wrong 2nd Slot offset for outputs
20/9 = More Net descriptor parameters required
20/10 = More Digital Node descriptor parameters required
20/11 = More Analog Node descriptor parameters required
20/12 = More Digital Slot descriptor parameters required
20/13 = More Analog Slot descriptor parameters required
20/14 = XML file configuration error
20/15 = Memory allocation error
20/16 = CobId duplicated or not defined
20/82 = XT210 I/O number doesn't match configured one
20/83 = XT210 Slot number doesn't match configured one
20/90 = No OS-Wire device connected
20/91 = Unknown OS-Wire I/O device connected
20/92 = OS-Wire I/O number doesn't match configured one
20/93 = OS-Wire Slot number doesn't match configured one

21/1 = Wrong process number

21/2 = Wrong number of parameters
21/3 = Function already defined
21/4 = Function not allowed
21/5 = Wrong semaphore number
21/6 = Send/Wait aborted for reboot
21/7 = Parameters out of range (send)
21/8 = Too many send variables
21/9 = Parameters out of range (wait)
21/10 = Decode function failed
21/11 = Encode function failed
21/12 = Parameters out of range (warning)
21/13 = SetTime function failed
21/14 = SetDate function failed
21/15 = Bit Overflow (outword)
21/16 = Bit Overflow (inword)
21/17 = Array too small for function
21/18 = String too long for function (Write Data)
21/19 = String too long for function (Read Data)
21/20 = Open file error
21/21 = Wrong type of variable
21/22 = Wrong type of operation
21/23 = Warning : too many variables
21/24 = Mismatch variables and file types
21/25 = Write file error
21/26 = Unknown variable type readed from file
21/27 = Wrong variable index readed from file
21/28 = Read file error
21/29 = End Of File
21/30 = Wrong filter function
21/31 = Wrong filter
21/32 = Wrong process number
21/33 = Process not found
21/34 = Axis not found
21/35 = Axis undefined
21/36 = Wrong number of slaves
21/37 = Wrong following modality
21/38 = Wrong block number (negative)
21/39 = Too many axis
21/40 = Request Denied from PLC Filters
21/41 = Wrong Line Search Modality
21/42 = Wrong Line Number (negative or null)
21/100 = Too many PLC tasks (Monitoring)
21/101 = Wrong command (Monitoring)
21/102 = Wrong variable index (Put/Get var)
21/103 = Wrong variable type (Put/Get var)
21/104 = List with different types of variables
21/105 = Error reading PLC Variables setup file
21/106 = Memory error loading PLC Variables
21/107 = PLC Variables setup file not found
21/108 = PLC Variable not found

21/110 = Error Reading WinPlus ASCII File
21/111 = Acknoledge variable is NOT a predefined variable
21/150 = Positive Limit
21/151 = Negative Limit
21/152 = Wrong translation error type
21/155 = User DLL function code out of range
21/156 = User DLL function block code out of range
21/157 = User DLL already loaded
21/158 = Range of User DLL function code overlaps existing functions
21/159 = Range of User DLL function block code overlaps existing Function blocks
21/160 = User DLL load error
21/161 = Too many User DLLs
21/162 = User DLL unload error
21/163 = Unload request of User DLL not loaded
21/168 = Error running program
21/169 = Timeout on program termination

22/1 = Too many Interpolators/Processes
22/3 = Too many Interpolator Activities
22/4 = Too many Movements
22/5 = Wrong number of axis
22/6 = Illegal movement (not exist)
22/7 = Illegal movement
22/8 = Illegal Intepolator/Process
22/9 = Memory allocation error
22/10 = Command not allowed
22/11 = Command not allowed in Continuous
22/12 = Program terminated
22/13 = Too many Interpolators
22/14 = Command not allowed in Abort Continuous
22/15 = Command not allowed in Terminate Motion
22/16 = Wrong Interpolator/Process PosMode parameter
22/17 = Wrong Interpolator/Process Type parameter
22/18 = Wrong Interpolator/Process ErrMode parameter
22/19 = Wrong Interpolator/Process EmgMode parameter
22/20 = Interpolator/Process not found
22/21 = Interpolator/Process already defined
22/22 = Interpolator/Process still working
22/23 = Movement is running
22/24 = Start denied
22/25 = Hold denied
22/26 = Exit from Hold denied
22/27 = Axis out of profile (restarting)
22/28 = Command not allowed for CNC
22/29 = Wrong FeedHold modality
22/30 = Unknown command class
22/31 = UnkNown command
22/32 = RoundOFF only for simple continuous movements
22/33 = Too many Continuous motion, Queue is full

22/34 = No Informations Available
22/35 = Spindle is not moving
22/36 = Spindle without ramps, function not available
22/37 = Tapping Length Error
22/38 = Tapping Dinamic Error
22/39 = Spindle axis cannot be programmed
22/40 = Axis not available, in use by normal movements
22/41 = Axis not available, in use by hold movements
22/42 = Axis with different clock
22/43 = Axis not defined
22/44 = Illegal axis Id
22/45 = Gantry axis cannot be programmed
22/46 = Axis not available
22/47 = Axis already defined into the command
22/48 = Axis not available, in use by Master/Slave
22/49 = Request available only from the Axis Owner
22/50 = Wrong Start condition for movement
22/51 = Wrong End condition for movement
22/52 = Wrong Break condition for movement
22/53 = Movement condition required
22/54 = Signal not available from Interpolator
22/55 = Wrong VelMode parameter
22/56 = Wrong Start Continuous modality
22/57 = Wrong End Continuous Tolerance management
22/58 = Wrong ChangePositionLoop modality
22/59 = Wrong Ramp modality
22/60 = Movement pass Positive Limit
22/61 = Movement pass Negative Limit
22/62 = Positive Limit reached during movement
22/63 = Negative Limit reached during movement
22/64 = Axis on Positive Overtravel
22/65 = Axis on Negative Overtravel
22/66 = Homing required
22/67 = Spindle Axis Required
22/68 = Spindle not Defined for Interpolator
22/69 = Gear Value Out of Range
22/70 = Null feed not allowed
22/71 = Jrk value (S ramp) out of range
22/72 = Variable not available from Interpolator
22/75 = Wrong Feed Modality
22/76 = Wrong SLL Value
22/77 = Wrong SSL Value
22/78 = SLL Value Greater Than SSL
22/79 = Function not available
22/80 = Wrong Origin number
22/81 = Origin not defined
22/82 = Wrong Probing Safety Distance Value
22/83 = Wrong Probing Approach Distance Value
22/84 = Wrong Probing Feed Value
22/85 = Probing Executed in Approach

22/86 = Probing not Executed
22/87 = Probing in Execution
22/88 = Probing not Required
22/89 = Wrong probe Radius
22/90 = Interpolator still running, request denied
22/92 = Wrong Circle Centre Modalty (ARM)
22/93 = Wrong Helix step
22/94 = Wrong Rollover Axis required position
22/95 = Wrong Fixed Cicle directions
22/96 = Wrong Fixed Cicle motion lengths
22/97 = Wrong Homing Direction
22/98 = Request denied Spindle is orienting
22/102 = Spindle Operatione Require Transducer
22/103 = Wrong Thread Length
22/104 = Wrong Thread Pitch Increment
22/105 = Wrong HandWheel Command
22/106 = Wrong HandWheel Scale Value
22/107 = Wrong HandWheel Pitch Value
22/108 = Wrong HandWheel Refresh Tick Value
22/109 = Wrong Spindle Use/Sharing Modality
22/110 = Spindle not Shared
22/111 = Spindle Request Denied While Interpolator is Running
22/112 = Spindle Already Used in Exclusive Modality
22/113 = Wrong Axis Use/Sharing Modality
22/114 = Request Denied While Axis is Moving
22/115 = Axis Already Used in Exclusive Modality
22/116 = Axis Request Denied While Interpolator is Running
22/117 = Request Denied Shared Axis Actived
22/118 = Axis not Shared
22/119 = Wrong Jerk Value
22/120 = Wrong Acceleration Value
22/121 = Wrong Deceleration Value
22/122 = Wrong Feed Value
22/123 = Wrong Dejerk Value
22/124 = Wrong Acceleration Time Value
22/125 = Wrong Deceleration Time Value
22/126 = Wrong Dynamic Override Modality
22/127 = Too many protected areas
22/128 = Undefined protected area
22/129 = Wrong Protected area command
22/130 = Protected area command not available (run mode)
22/131 = Too many programmed axes
22/150 = Positive Limit
22/151 = Negative Limit
22/152 = Protect Area Limit
22/201 = Master/Slave setup mismatch
22/202 = Not slave axis required
22/203 = Slave axis still linked with a Master
22/204 = Slave axis commands pending
22/205 = Wrong following Modality

22/206 = Wrong Master/Slave activation or deactivation
22/207 = Reset quote for Master differ from Slave axis quote
22/208 = Reset quote for Master must be 0
22/209 = Illegal Use of Slave Axis
22/210 = Illegal Use of Master Axis
22/211 = Illegal Use of Follower Axis
22/212 = Illegal Spline Init Position
22/213 = Illegal TCP request
22/214 = TCP Axes Number Different from Kinematics
22/215 = TCP Axes Id Different from Kinematics
22/216 = Axis Type not Virtual
22/217 = Homing Request Denied TCP Activated
22/218 = Homing Request Denied for Virtual Axes
22/219 = Virtual Axis not Available
22/220 = TCP change Not Available with Realtime Tool Compensation
22/221 = TCP Virtual and Rotary axis cannot be Moved Together
22/222 = Slave Axis is cannot be a Virtual Axis
22/300 = Process not defined
22/301 = LookAhead already started
22/302 = LookAhead not started
22/303 = Cycle start denied
22/304 = Cycle stop denied
22/305 = Point to Point Movement required into continuous
22/306 = Function not available for PLC Process
22/307 = Information Disalignment Axis Number (CNC)
22/308 = Information Disalignment Axis Id (CNC)
22/309 = Function not available during UPR
22/320 = Open Cam file error
22/321 = LF not found (0x0a) at the end of file
22/322 = Cam file does not exist
22/323 = Cam table not strictly monotonic
22/324 = Error first record of cam file
22/325 = Error record of cam file
22/326 = Max cam number reached
22/327 = Max action number reached
22/328 = Invalid cam id
22/329 = Invalid action id
22/330 = Cam not defined
22/331 = Invalid cam rollover pitch
22/332 = Cam active
22/333 = Disengage position greater than cam rollover pitch
22/6001 = Axis %2 (Id %1) homed%nMicro-Marker distance %3

23/1 = Syntax error on line:%2%n block: %1
23/2 = Open file NMC_BINARY.DAT error
23/3 = Open file LDIRDATA.DAT error
23/4 = Insufficient memory on process
23/5 = Process undefined
23/6 = Format error
23/7 = Wrong variable identifier

23/8 = Wrong axis name
23/9 = Too many variables in RD/WR
23/10 = Wrong axes number into the command
23/11 = Error reading value variable
23/12 = Wrong data into ODM configuration file
23/13 = TCP memory mapping file error
23/14 = Error generating TCP restore file
23/15 = Invalid value for variable
23/16 = Command unknown
23/17 = Command and system status not congruent
23/18 = Wrong parameter command
23/19 = Internal error : nc message error
23/20 = RCM not available
23/21 = Too many virtual axis requested
23/50 = Wrong number of axes for G code
23/51 = Fixed cycle parameters missing
23/52 = Missing parameters for G code
23/53 = Missing J and/or K for G83 cycle
23/54 = Missing I and/or J for G2/G3/G12/G13/G14 code
23/55 = Probing cycle parameters missing
23/56 = Undefined variable
23/57 = Expression Overflow
23/58 = Function not allowed
23/59 = Illegal use of slave axis
23/60 = Operand not allowed in fixed cycle
23/61 = K parameter not allowed in G84
23/62 = Wrong programming of G2/G3 code
23/63 = Illegal number of operands
23/64 = Illegal number of pseudo axis
23/65 = Illegal number of axes in G33 code
23/67 = G not allowed
23/68 = Operand not allowed with G code
23/69 = Block and system status not congruent
23/70 = G and program state not congruent
23/71 = Movement index not congruent
23/72 = Programming type not congruent
23/73 = Current programming not congruent
23/74 = G and dynamic mode not congruent
23/75 = G41/G42 and part program status not congruent
23/76 = G needs spindle with transducer
23/77 = G not congruent with feedrate mode
23/78 = Operand and part program status not congruent
23/79 = PLC operand and part program status not congruent
23/80 = M and dynamic mode not congruent
23/81 = M/T/S and motion type not congruent
23/82 = Probing cycle operands inhibited
23/83 = Missing third axis for helix
23/84 = "Expedite" function without motion
23/85 = Feed not programmed
23/86 = Speed not programmed

23/87 = Read only variable
23/88 = Part program block not allowed from serial line
23/91 = Nesting of IF greater than 32
23/92 = ELSE not allowed
23/93 = ENDIF not allowed
23/94 = Part program record too long%n(Program: %1 Line: %2)
23/95 = Part program %1 access denied
23/96 = Program already selected
23/97 = Part Program name too long (>127 char)
23/98 = DRIVE not found
23/99 = Path specified not found %n%1
23/100 = File %1 not found
23/101 = Illegal file name
23/102 = File access error
23/103 = Error during part program file handling
23/104 = Activation part program denied
23/105 = Deactivation part program denied
23/106 = Part program not selected
23/107 = Tool direction and plane compensation not congruent
23/108 = Repeated axes in UDA/SDA command
23/109 = Use of illegal axis in safety area definition
23/110 = Triletterals not allowed
23/111 = M code not allowed
23/112 = Repeated length or diameter in RQT/RQP
23/113 = G and Spline status not congruent
23/114 = PLC operand and Spline status not congruent
23/115 = Checkpoint with no K parameter defined
23/116 = Parameter axis missing from computation axis list
23/117 = Programmed axis does not belong to Spline
23/118 = Error acceding to saved paramacro
23/119 = Error loading file %1 at line %2
23/120 = Wrong TCP for Spline transformation
23/121 = Missing theoretical or measured points for G57
23/122 = Erroneous points for G55 and G56
23/123 = Record parameter specified for R/W an ASCII file
23/124 = Erroneous size for serial terminator
23/125 = Missing or too many data in $PTECH loading%n(File: %1 Line: %2)
23/126 = Data or line format erroneous in $PTECH loading%n(File: %1 Line: %2)
23/150 = Illegal argument for TAN
23/151 = Illegal argument for SQR
23/152 = Too many programmed axes
23/153 = Division by zero
23/154 = String too long
23/155 = Label duplicated
23/156 = Undefined label
23/157 = Label %1 too long
23/158 = Program table overflow
23/159 = Label table overflow
23/160 = End of program
23/161 = LF not found (0x0a) at the end of program

23/162 = Beginning of program
23/163 = Nesting of RPT greater than 5
23/164 = Nesting of subroutine greater than 7
23/165 = Nesting of EPP greater than 5
23/166 = RPT/EPP/IF/ELSE cycle open at end of program
23/167 = ERP without RPT
23/168 = Modal paramacro already active
23/169 = Paramacro not configured
23/170 = Illegal argument for logarithm function
23/173 = Too many elements in safety area definition
23/174 = Safety area could not be closed(elements overflow)
23/175 = Active area can not be redefined
23/176 = Error in limits for 3D safety area
23/177 = XMLrules.txt not loaded. XML commands not executed
23/178 = Syntax error in XMLrules.txt
23/179 = High Speed Machining function not available
23/180 = Interpolation Type not congruent with HSM
23/181 = High Speed Machining disabled from ODM
23/182 = XMLrules.txt is updating...execution denied
23/183 = Internal error during through mode
23/184 = Full data buffeer (through mode)
23/189 = Local alias duplicated on block : %n%1
23/190 = Local alias too long or incorrectly defined on block : %n%1
23/191 = Too many local aliases defined on block : %n%1
23/192 = Undefined alias on block : %n%1
23/250 = Axis not referenced
23/251 = Undefined DPT for probing cycle
23/252 = Too many "expedite" M codes
23/253 = Undefined M code
23/254 = Circle not congruent
23/255 = Wrong threading parameters (I,K,R)
23/256 = Helix pitch not congruent
23/257 = Axes of plane need same scale factor
23/258 = Geometry not congruent
23/259 = Profile not congruent
23/260 = Wrong direction on profile
23/261 = Error disabling cutter compensation
23/262 = Full transit buffer
23/263 = Too many blocks to resolve
23/264 = Entry in safety zone
23/265 = Fixed cycle on rotate plane
23/266 = Fixed cycle length data not congruent
23/267 = Fixed cycle data not congruent
23/268 = Fixed cycle not allowed
23/269 = Wrong probing cycle programming
23/270 = Positive software overtravel limit
23/271 = Negative software overtravel limit
23/272 = Axes not on profile
23/273 = Error in exit HOLD : mode changed
23/274 = Block not allowed in HOLD

23/275 = Active reset denied
23/276 = Wrong use of rollover axis with G90
23/277 = Wrong use of real axis during a virtualization modality
23/278 = Tool Direction active : movement not allowed
23/279 = Command not allowed during search in memory
23/280 = Geometric out buffer full
23/281 = Axis shared with PLC
23/282 = Safety area is not convex
23/283 = Safety area not defined.
23/284 = Traslation on axis not on safety area
23/285 = Axis type not congruent for axis tangential control
23/286 = Automatic return to profile not allowed after MDI in HOLD
23/287 = Programmed a circular radius on a concave angle
23/288 = Tool Center Point disabled from ODM
23/289 = More than one offset assigned to an axis
23/290 = Requested XDA command with no master / slave associated
23/291 = Real axis programmed while virtualization running
23/292 = Access at disabled line of $PTECH table
23/293 = Cam status is inconsistent with the request
23/350 = Mode to select out of range
23/351 = Axes number to select out of range
23/352 = Too many axes selected for manual move
23/353 = Bad select mode for cycle
23/354 = Data length out of range
23/355 = Operative limit definition wrong
23/356 = Offset length not defined for the axis
23/357 = Tool orientation code wrong
23/358 = Error from servo environment
23/359 = Manual movement not executed: no axes configured
23/360 = Axis not configured for GTA/GTS/TCP
23/361 = PLC axis in GTA command
23/362 = GTA/GTS/AX_SHARE: axis/spindle not available
23/363 = Id duplicated in request
23/364 = Request Id identifies splindle
23/365 = Too many required tools
23/366 = Start index for tools search out of range
23/367 = Range of tools search out of range
23/368 = Axis ID out of range
23/369 = Offset length assigned on multiple axis
23/370 = Offset length out of range
23/371 = Duplicated axis name
23/372 = Unrecognized CAM command
23/400 = Set spindle speed failed
23/401 = New tool request failed
23/402 = M execution failed
23/403 = Pseudo axes programming failed
23/404 = Axis motion inhibited
23/405 = End of move failed
23/406 = Too many blocks without movement in continuos mode
23/450 = Axis on profile

23/451 = End of automatic return to profile
23/452 = End of multi block retrace
23/453 = End of search in memory
23/500 = Actual process number programmed
23/501 = ASSET option not installed
23/502 = Circles/lines not defined
23/503 = Fixed Cycle not possible with shared spindle
23/504 = Spindle required with G84 or G86
23/505 = Dry Run already activated
23/506 = Requested process does not exist
23/507 = Wrong process type
23/508 = Full queue on target process
23/509 = Data sent too long
23/510 = Data loading failed
23/511 = Message already exist in queue
23/512 = EXE or ECM failed
23/513 = Channel already in use
23/514 = Error opening file
23/515 = File already open
23/516 = File opened for reading does not exist
23/517 = Logic error acceding to file
23/518 = Erroneous access type on file
23/519 = End of file reached
23/521 = Missing path for file
23/522 = Error trasmitting data
23/525 = Binary file is corrupted
23/526 = Acceding to binary file with erroneous record type
23/527 = Record too long writing on binary file
23/537 = Table already locked
23/539 = Serial port not connected
23/540 = Timeout trasmitting data
23/541 = Timeout receiving data
23/542 = Error on serial line
23/543 = Missing data in reception
23/553 = Error in UPR programming
23/554 = Incremental UPR not allowed
23/555 = Dual axis and UVP not compatible
23/556 = Dual axis and UVA not compatible
23/557 = Invalid cylindrical radius value
23/558 = Dual axis and UVC not compatible
23/559 = Kynematic Id out of range
23/560 = Function not available for current TCP type
23/561 = Slave axis in TCP configuration
23/562 = Error in TCP configuration
23/563 = Error in getting/releasing axis
23/564 = UPR and TCP configuration conflict
23/565 = Null norm for required unit vectors
23/566 = Error in programming unit vectors
23/568 = p,q,d with no m,n,o specified
23/569 = Kynematics Id needed for TCP activation

23/570 = Name for tool direction axis already in use
23/571 = Function not available
23/572 = tcp type is not compatible with running TCP
23/573 = Virtual axis cannot be added
23/700 = Unrecognized command for $PTECH loading
23/701 = Wrong number of data sent to $PTECH
23/800 = NCUSER error - (%1)
23/801 = NCUSER warning - (%1)
23/900 = Preload table is full
23/901 = File already in preload table
23/902 = Swap / remove of a file not in preload table

24/1 = Command rejected
24/2 = Command failed
24/3 = Invalid node Id
24/4 = Drive busy
24/5 = Invalid parameter
24/6 = Trasmission failure
24/7 = DualPort memory access error
24/8 = RTC Init timeout

25/1 = Memory error for mapping system history files
25/2 = Error creating system history files
25/3 = Error reading system history files
25/4 = Wrong system history file type
25/10 = Memory error for mapping table files
25/11 = Error creating table files
25/12 = Error reading table files
25/13 = Error writing table files
25/14 = Wrong table type
25/15 = Wrong table field number
25/16 = Wrong table field type
25/17 = Table is locked
25/18 = Error restoring table files
25/19 = Error backuping table files
25/20 = Wrong event process
25/21 = Wrong event message type
25/22 = Insufficient memory for reading event message
25/30 = Error saving Retain Memory (file)
25/31 = Retain Memory corrupted
25/32 = Error restoring Retain Memory (file)
25/33 = Memory allocation error restoring Retain Memory
25/34 = Retain Memory image (file) corrupted

26/1 = Circles/lines not intersecting
26/2 = Coinciding circles/lines/points
26/3 = Coincident or internal circles
26/4 = Point inside circle
26/5 = Parallel lines
26/6 = Wrong definition of circles/lines

26/7 = Aligned points
26/8 = Circles/lines not defined
26/9 = Not parallel circles/lines/points
26/10 = Different radius between initial-final points
26/11 = Circle not congruent
26/12 = Normal vector not defined

27/1 = Function not yet implemented
27/2 = Invalid environmente Id
27/3 = Reached max number of environment
27/4 = Environment Id already defined
27/5 = Undefined environmentId
27/6 = Failed memory allocation for environment
27/7 = Unrecognized command
27/8 = Invalid synchronization mode
27/9 = Communication message too long
27/10 = Failed segment allocation memory
27/11 = Only one point specified
27/12 = Error opening debug file
27/13 = Error in Spline configuration
27/14 = Error in axis identification
27/15 = Failed saving of InfoAgg
27/16 = Error in createSplineCurve
27/17 = Error in reduceSplineOscillations
27/18 = Error handleSharpCornersInSplineCurve
27/19 = Error in chop_off_spline_right_end
27/20 = Error in chop_off_spline_left_end
27/21 = Error in merge_spline_segments_via_transition_zone
27/22 = Error in fitPrevCurveSegmentToCurrent
27/23 = Error in chopOffSplineCurves
27/24 = Error in calcQuinticPolynomial
27/25 = Error in memory allocation
27/26 = Error axis configurazion
27/27 = Errore in search of InfoAgg

31/1 = System Configuration not found
31/100 = Internal Error Booting IPC
31/101 = Internal error Boot not available

32/1 = Insufficient memory
32/2 = Insufficient MBOX channels
32/3 = Too many requests
32/4 = Resource id out of range
32/5 = Reserved
32/6 = Axis not found
32/7 = Primary axes board not found
32/8 = Abnormal axis configuration file
32/9 = Abnormal extended data configuration file
32/10 = File not found
32/13 = Digital axis required

32/14 = Analog axis required
32/15 = Invalid request
32/16 = Invalid resource type
32/17 = Resource already exists
32/18 = Resource not available
32/19 = Transducer not found
32/20 = Converter not found
32/21 = Axes bootstrap halted
32/22 = Command already active
32/23 = Command not possible
32/24 = Command timeout
32/25 = Invalid clock value
32/32 = Monitoring already active
32/33 = Monitoring not atctivated
32/34 = Monitoring function not exists
32/35 = Variable not configured
32/36 = Incorrect number of variables
32/37 = Variable value out of range
32/48 = Error on gantry master slave definition
32/49 = Operation not possible on gantry slave
32/50 = Function allowed only on gantry axes
32/51 = Gantry axes in skew error
32/52 = Skew offset conversion error
32/53 = Function not allowed on gantry axes
32/64 = Internal error 0
32/65 = Internal error 1
32/66 = Internal error 2
32/67 = Internal error 3
32/68 = Internal error 4
32/69 = Internal error 5
32/70 = Internal error 6
32/71 = Internal error 7
32/72 = Internal error 8
32/73 = Internal error 9
32/80 = Boot error phase 0
32/81 = Boot error phase 1
32/82 = Boot error phase 2
32/83 = Boot error phase 3
32/84 = Boot error phase 4
32/85 = Boot error phase 5
32/86 = Boot error phase 6
32/87 = Boot error phase 7
32/88 = Boot error phase 8
32/96 = Invalid position error type
32/97 = Entity queue full
32/98 = Interpolation active
32/99 = Invalid servo type
32/100 = Probe anomaly
32/101 = Filter not configured
32/102 = Filter id not valid

32/103 = Filter command not valid
32/112 = Communication timeout
32/113 = Communication channel busy
32/114 = Axis inhibited for failure
32/115 = Invalid communication phase
32/116 = Node not found on bus
32/117 = Channel inhibited for failure
32/128 = Invalid parameter id
32/129 = Axes id mismatch
32/130 = KCKV conversion error
32/131 = Space tolerance conversion error
32/132 = Value out of range
32/133 = Electrical pitch conversion error
32/134 = Transducer not configured
32/135 = Illegal gear number
32/136 = Digital axis not available for exchange
32/137 = Axis in exclusive sharing mode
32/138 = Invalid axes board
32/139 = Max velocity too high
32/140 = Invalid parameter type
32/147 = Error opening ALT file
32/148 = Error opening APL file
32/149 = Error parsing ALT/APL files
32/150 = ALT not completed
32/151 = Error reading compensation data from file
32/152 = Compensation type not implemented
32/155 = Check drive parameters is not done
32/160 = Standstill error threshold exceeded
32/161 = Following error threshold exceeded
32/162 = Timeout entering in tolerance
32/163 = Tolerance failure
32/164 = Marker too far
32/165 = Broken wire detection
32/176 = Protocol error during CP0
32/177 = Protocol error during CP1
32/178 = Protocol error during CP2
32/179 = Protocol error during CP3
32/180 = Protocol error during CP4
32/181 = Protocol timeout error
32/182 = Protocol not available
32/192 = XML config file error
32/193 = Data too big into XML config file
32/194 = Missing data into XML config file
32/195 = Drive identification mismatch into XML config file
32/196 = EtherCAT configured resources require XML configuration file

33/1 = Overflow in read/write OS3's dualport
33/2 = OS3 HW interface not found
33/3 = Axis not found on OS-Wire bus
33/4 = OS3 flash file error

33/5 = Error during OS3 firmware upgrading
33/6 = OS3 write parameter error
33/7 = Too much read/write parameters
33/8 = Max torque = 0
33/9 = Amotmax = 0
33/10 = OS3 velocity loop calculation error
33/11 = OS3 tuning calculation error
33/12 = OS3 drive size mismatch
33/13 = OS3 Motor part number mismatch
33/14 = OS3 parameters congruence test not possible
33/15 = OS3 illegal request
33/160 = Drive OS3

// OS3 Command errors
33/257 = E01 Command cannot be executed
33/258 = E02 Monitoring table overflow
33/259 = E03 Monitoring already active
33/260 = E04 No variable to be monitored
33/261 = E05 Error during transmission
33/262 = E06 Timeout error during transmission
33/263 = E07 Timeout error during reception
33/264 = E08 Checksum error
33/265 = E09 Echo error
33/266 = E10 Hyperface error
33/267 = E11 Feedback table overflow
33/268 = E12 Variable ID wrong
33/269 = E13 Parameters table overflow
33/270 = E14 Parameter ID wrong
33/271 = E15 Parameter write inconsistent
33/272 = E16 Invalid memory address
33/273 = E17 Inconsistent data in Sequencer
33/274 = E18 Invalid command parameters
33/275 = E19 Point2Point command unavailable
33/276 = E20 Offset identification not available
33/277 = E21 Synchronisation timeout on RTC2
33/278 = E22 FPGA register ID wrong
33/279 = E23 Number of locations overflow
33/280 = E24 Qep counter release timeout
33/281 = E25 External Td Qep counter release timeout
33/282 = E26 No encoder interface connected
33/283 = E27 Power error

34/1 = Node not found
34/2 = Invalid buffer
34/3 = IDN element number not valid
34/4 = No data from SVC
34/5 = SVC data truncated
34/6 = Error from command IDN S-0-0146
34/7 = Timeout during drive enable
34/8 = Invalid IDN type

34/15 = Invalid request
34/32 = Command acknowledgment error from IDN S-0-0146
34/33 = Command acknowledgment error from IDN P-0-0014
34/34 = Command acknowledgment error from IDN S-0-0172
34/35 = Command acknowledgment error from IDN S-0-0170
34/36 = Error writing IDN S-0-0405
34/160 = Drive SERCOS

35/3 = CAN Node not found
35/4 = CAN write parameter error
35/5 = CAN Drive not configured
35/6 = Communication Error
35/160 = Drive CAN

36/1 = Hardware Tip Touch Emergency
36/2 = Software Quick Tip Touch Emergency
36/3 = Software Normal Tip Touch Emergency
36/4 = Invalid Measurement Emergency
36/5 = Avoid Hole Emergency

37/1 = EtherCAT node not found

38/1 = Mechatrolink node not found
38/2 = Error during SENS_ON command execution
38/3 = Error during SYNC_SET command execution
38/4 = Error during SMON command execution
38/5 = WDT error
38/6 = Error during command execution
38/7 = Timeout error command execution
38/8 = Warning while reading/writing drive parameters
38/9 = Different drive parameters from NC database
38/10 = Alarm during boot of drive
38/11 = Mechatrolink communication error
38/160 = Drive Mechatrolink

49/1 = Memory allocation error
49/2 = Invalid serial line number
49/3 = Serial line not present
49/4 = Invalid request
49/5 = Invalid call modality
49/6 = Frame too big
49/16 = Invalid baudrate configuration
49/17 = Invalid byte size configuration
49/18 = Invalid parity configuration
49/19 = Invalid flow control configuration
49/20 = Invalid protocol configuration
49/21 = Invalid bitstop configuration
49/32 = Error opening device
49/33 = Error closing device
49/34 = Error reading DCB

49/35 = Error writing DCB
49/36 = Error reading timeouts
49/37 = Error writing timeouts
49/38 = Transmission error
49/39 = Error during purge
49/40 = Error during setcommmask
49/41 = Error during escape
49/42 = Error during clearcomm
49/43 = Error during readfile
49/48 = No more Tx/Rx blocks
49/49 = Not all characters transmitted
49/50 = Timeout at receipt beginning
49/51 = Overrun during receipt
49/52 = Frame error
49/53 = Parity error
49/54 = Overrun receipt in O.S.
49/55 = Mismatch number of read characters
49/240 = Internal error 1
49/241 = Internal error 2

50/1 = Invalid Option code
50/2 = Wrong Primary board
50/3 = Missing osypdata file
50/4 = Incorrect PAK
50/5 = PAK not installed
50/6 = Error during FPGA download
50/7 = Board registry delete error
50/8 = Options A10/A11 not enabled with "EXPORT" board
50/10 = Option not enabled (A22 Tool magazine)
50/11 = Option not enabled (A23 Open SDK User DLL)
50/13 = Option not enabled (A24 Manual Digitizing)
50/17 = Option not enabled (A25 Cam Editor)
50/19 = Option not enabled (A26 Remote Diagnostic)
50/20 = Option not enabled (A27 Laser Digitizing)
50/22 = Option not enabled (A02 Layout Builder)
50/23 = Option not enabled (A03 Process Controller + Table Editor, System History, Machine Plot)
50/24 = Option not enabled (A06 Cndex Link Communication)
50/27 = Option not enabled (A10 3 Axis interpolation limit)
50/28 = Option not enabled (A11 4 Axis interpolation limit)
50/30 = Option not enabled (A14 Bidirectional Linear Compensation)
50/31 = Option not enabled (A15 Straightness-Squareness Compensation)
50/32 = Option not enabled (A16 Volumetric Compensation)
50/33 = Option not enabled (A53 Canopen)
50/34 = Option not enabled (A54 Profibus)
50/35 = Option not enabled (A55 Ethercat)
50/36 = Option not enabled (A56 Ethercat Dual channel/Ring topology)
50/37 = Option not enabled (A57 Ethercat Hot Plug)
50/41 = The number of axes/transducers configured exceeds the maximum number allowed by PAK
50/49 = Option not enabled (A05 PathView)
50/50 = Option not enabled (A12 SawCad)

50/51 = Option not enabled (A13 SketchCad)
50/52 = Option not enabled (A20 Conversational for mill)
50/53 = Option not enabled (A21 Conversational for lathe)
50/58 = The number of processes configured exceeds the maximum number allowed by PAK
50/59 = Option not enabled (A30 UPR Enable)
50/60 = Option not enabled (A31 TCP Enable)
50/61 = Option not enabled (A32 Extended TCP Enable)
50/62 = Option not enabled (A33 High Speed Enable)
50/63 = Option not enabled (A34 Axis and Spindle Migration)
50/64 = Option not enabled (A35 Dual Axes UDA/SDA)
50/65 = Option not enabled (A36 Extended Dual Axes XDA)
50/66 = Option not enabled (A37 Following Axes AXF)
50/67 = Option not enabled (A38 Splines(G61) Enable)
50/68 = Option not enabled (A39 Workpiece setting error compensation)
50/69 = Option not enabled (A40 Extended Workpiece setting error compensation)
50/70 = Option not enabled (A41 Electronic CAM)
50/75 = Option not enabled (A42 XML from Part Program)
50/77 = Option not enabled (A43 Access to Table from ASSET)
50/78 = Option not enabled (A44 Multi-process synchronization from ASSET)
50/79 = Option not enabled (A45 TCP/IP Communication)
50/80 = Option not enabled (A46 XML file access from Asset)
50/81 = Option not enabled (A47 RS232from Asset)
50/83 = Option not enabled (A17 Process Controller Multi CNC Connection)
50/84 = Option not enabled (A18 A18 Table Editor Bulider)
50/91 = Option not enabled (A48 Motion Library Multi-Axes Interpolation)
50/92 = Option not enabled (A49 Controls Library)
50/93 = Option not enabled (A50 Custom Database)
50/94 = Option not enabled (A51 I/O Remapping)

51/32769 = Memory allocation error
51/32770 = Invalid configuration
51/32771 = Invalid execution mode
51/32772 = Board not found
51/32773 = Parameter out of range
51/32774 = Hardware not found
51/32784 = Error while reading diagnostic log
51/32785 = No SoE server available
51/32800 = Bus in NRT phase
51/32801 = Error during communication phase changing
51/32816 = Node not found on bus
51/32817 = Comunication channel busy
51/32818 = Comunication channel in not recoverable error
51/32819 = Invalid IDN request type
51/32820 = Communication timeout
51/32821 = Negative ack from command IDN
51/32832 = Service channel internal error
51/32833 = Big data for service channel
51/32834 = Invalid IDN element
51/32835 = Data read truncated

52/1 = Memory allocation error
52/2 = Error while DLL loading
52/3 = No more descriptor
52/4 = Invalid board
52/5 = Function not available
52/6 = Invalid parameter
52/7 = Node not found
52/4097 = Error opening drive
52/4098 = Error boards listing
52/4099 = Error opening channel
52/4100 = Error opening sys device
52/4101 = Error reset sys device
52/4102 = Error bus on
52/4103 = Error opening communication channel
52/4104 = Error set host state
52/4105 = Invalid board
52/4106 = Error reading mailbox state
52/4107 = Error packet reading
52/4108 = Error packet writing
52/4109 = Error reading common state
52/4110 = Error reading channel information
52/4111 = Error closing sys device
52/4112 = Data exchange error
52/4113 = Protocol not loaded
52/4115 = Configuration file not found
52/4116 = Error opening configuration file
52/4117 = Error reading configuration file
52/4118 = Error loading configuration
52/4119 = Reset channel error
52/4120 = Netx not ready
52/4121 = Flag COS not set

53/1 = CanOpen PutMessage timeout
53/2 = CanOpen GetMessage timeout
53/3 = CanOpen Abort SDO
53/4 = CanOpen Invalid command
53/5 = CanOpen Send mailbox is full
53/6 = CanOpen Number node incongruent
53/11 = CanOpen Node is not present
53/12 = CanOpen Node is not in operational state, SDO channel busy
53/13 = CanOpen Node request diagnostic
53/31 = CanOpen DRIVER Board not initialized
53/32 = CanOpen DRIVER Error in internal init state
53/33 = CanOpen DRIVER Error in internal read state
53/34 = CanOpen DRIVER Command on this channel is active
53/35 = CanOpen DRIVER Unknown parameter in function occured
53/36 = CanOpen DRIVER Version is incompatible with DLL
53/40 = CanOpen DEVICE Dual port ram (board not found)
53/41 = CanOpen DEVICE Not ready (ready flag failed)

53/42 = CanOpen DEVICE Not running (running flag failed)
53/46 = CanOpen DEVICE Send mailbox is full
53/47 = CanOpen DEVICE PutMessage timeout
53/48 = CanOpen DEVICE GetMessage timeout
53/49 = CanOpen DEVICE No message available
53/52 = CanOpen DEVICE IO data exchange failed
53/53 = CanOpen DEVICE IO data exchange timeout
53/88 = CanOpen DRIVER General error
53/90 = CanOpen DRIVER I/O WatchDog failed
53/91 = CanOpen DRIVER Device WatchDog failed
53/140 = CanOpen SYNC Code is full
53/141 = CanOpen File .dbm board 0 not found
53/142 = CanOpen File .dbm board 1 not found
53/150 = Node not found on DBM
53/151 = Input not found on DBM
53/152 = Output not found on DBM
53/153 = DBM file not found
53/154 = File WINPLUS.bus empty
53/155 = DBM offset out of range

54/1 = Memory allocation error
54/2 = Parameter out of range
54/3 = EtherCAT master not allocated
54/4 = EtherCAT master already allocated
54/5 = Error during init EtherCAT master
54/6 = Error during deinit EtherCAT master
54/7 = Error during DCM enable
54/8 = Communication phase change error
54/9 = Error of Link Layer instance
54/10 = EtherCAT master configuration error
54/11 = Configuration file not found
54/12 = Error during EtherCAT master registration
54/13 = DCM configuration error
54/14 = DCM initialisation error
54/15 = Error during DCM set value
54/16 = Communication phase request not valid
54/17 = Error while reading node's offset
54/18 = Node not configured or not found on bus
54/19 = Mailbox error request
54/20 = No diagnostic informations available
54/21 = Lost diagnostic informations
54/22 = Invalid axes board
54/23 = PTS activation error

56/1 = XML file invalid path
56/2 = XML file invalid handle
56/3 = XML file does not exists
56/4 = XML invalid mode
56/5 = Cannot allocate memory for XML file
56/6 = XML filename too long

56/7 = XML file load error
56/8 = Cannot save XML readonly file
56/9 = Cannot save XML file
56/10 = Element not found in XML file
56/11 = Cannot create path XML file
56/12 = Cannot convert type in XML file
56/13 = Too many XML files
56/14 = Var not found in XML file
56/15 = Internal error in XML management
56/16 = Structure not supported in XML file
56/17 = XML attribute not found

59/8 = Unrecognized Euler rotation
59/9 = Error computing Euler angles
59/10 = Error computing vector norm

61/1 = Incorrect communication channel initialisation
61/2 = Invalid protocol selection
61/3 = Invalid buffer mode selection
61/4 = Invalid user parmeter
61/5 = JL-080 reset failed
61/6 = Communication mode setting timeout
61/7 = Communication parameter setting timeout
61/8 = Initialisation sequence and process timeout
61/9 = Insufficient JL-080 RAM
61/10 = JL-080/JL-100 RAM check error
61/11 = Too many drives for the communication cycle time
61/12 = Slave address out of range
61/13 = Impossible to switch the buffer
61/14 = Data not received or corrupted
61/15 = Synch not received
61/16 = No synch
61/17 = JL-080 system error
61/18 = JL-080 application error
61/19 = Memory allocation error
61/32 = JL-100 not ready
61/33 = Timeout while detecting slaves
61/34 = JL-100 application error

// Emergency messages
[EMERGENCY]

1 = Encoder fault%nAxis %2 (Id %1)%nCause : %3
2 = Anomaly coming from servo%nCause : %3
20 = CAN Communication error%nNode %1 Board %2 Type %3
30 = OS3 I/O Watch Dog%nNode %1 Type %2
31 = OS-Wire Communication error%nNode %1
32 = Drive fault%nAxis %2 (Id %1)%nCause : %3
80 = System Watch Dog
81 = Power Fail

82 = 4Control exception%nTask %1 for %2
83 = PLD Watch Dog
90 = ShutDown
98 = Software System Anomaly
99 = System Exception%nCause : %1
101 = Servo error Axis %2 (Id %1)%nCause : %3%nValue : %4
102 = Skew error%nAxis %2 (Id %1)%nCause : %3
103 = Tolerance error Axis %2 (Id %1)%nCause : %3
104 = Marker too far%nAxis %2 (Id %1)%nCause : %3
105 = Axis initialisation error%nAxis %2 (Id %1)%nCause : %3
106 = Axis configuration error%nAxis %2 (Id %1)%nCause : %3
107 = Command error on Axis %2 (Id %1)%nCausa : %3
120 = Exit from SW operative range%nAxis %2 (Id %1) for %3
121 = Enter in SW operative range%nAxis %2 (Id %1) for %3
122 = Exit from HW operative range%nAxis %2 (Id %1) for %3
123 = Enter in HW operative range%nAxis %2 (Id %1) for %3
130 = Drive initialisation error%nAxis %2 (Id %1)%nCause : %3
131 = Drive command error%nAxis %2 (Id %1)%nCause : %3
132 = Drive alarm on Axis %2 (Id %1)%nSource : %3%nCode : %4
133 = Drive communication error%nAxis %2 (Id %1)%nCause : %3
140 = Hermes communication error
141 = Sensor Error: Hardware Nozzle Touch
142 = Sensor Error: Quick Software Nozzle Touch
143 = Sensor Error: Normal Software Nozzle Touch
144 = Sensor Error: Tip fault
180 = Motion Emergency on Axis %2 (Id %1)%n%3
181 = Motion Emergency on Task %1%n%2
190 = Emergency Switch Opened (PLC)
191 = Emergency Switch Opened (System)
201 = CANOpen I/O Warning%nNode %1 Board %2 Type %3
202 = SERCOS Warning%nScheda %1
204 = CAN Vector I/O Warning%nNode %1 Board %2 Type %3
206 = PROFIBUS Master I/O Warning%nNode %1 Board %2 Type %3
207 = PROFIBUS Slave I/O Warning%nNode %1 Board %2 Type %3

Contacts:
PRIMA ELECTRO S.p.A.
Strada Carignano, 48/2
Moncalieri (TO) - ITALY
Tel. +39 011 9899 800
Web: www.primaelectro.com
e-mail: sales@primaelectro.com

Copyright © 2013 by PRIMA ELECTRO S.p.A.